



**UniMAP**

**UNIVERSITI MALAYSIA PERLIS**

COURSE NAME	<b>ENGINEERING SKILLS</b>		
COURSE CODE	<b>ECT 111/3</b>	LAB NO.	

## **LAB MODULE**

### **CO4: AUTOCAD**

#### **LEVEL OF COMPLEXITY**

1	2	3	4	5	6
KNOWLEDGE	COMPREHENSION	APPLICATION	ANALYSIS	EVALUATION	SYNTHESIS
√	√	√	√		

**ENGINEERING & INNOVATION  
CENTRE**

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# TUTORIAL 1: Creating A Titleblock

## OBJECTIVES

At the end of this session you will be able to:

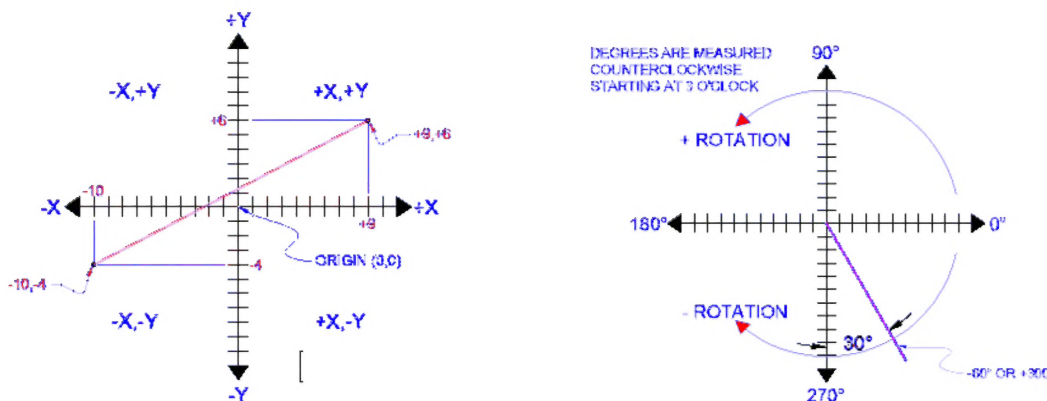
- 1) Familiarize with AutoCAD user interface, commands and basic drafting tools;
- 2) Open a blank AutoCAD file, create and save a drawing template;
- 3) Create a Titleblock using coordinate and basic drafting tools;
- 4) Create new layers and add text to a drawing; and
- 5) Insert a Block into Layout (Paper) Space;

**AutoCAD**<sup>®</sup> is a general purpose computer aided design (CAD) software which you can use to prepare a wide variety of two-dimensional drawings and three –dimensional models. AutoCAD brings the sophisticated technology, previously available only on large and costly systems, to the desktop and laptop user.

Virtually there is no limits to the kinds of the drawing that you can prepare using **AutoCAD**<sup>®</sup>, if a drawing can be created by hand it also can be generated using AutoCAD.

### 1.1 The AutoCAD Coordinate System and Angular Measurement

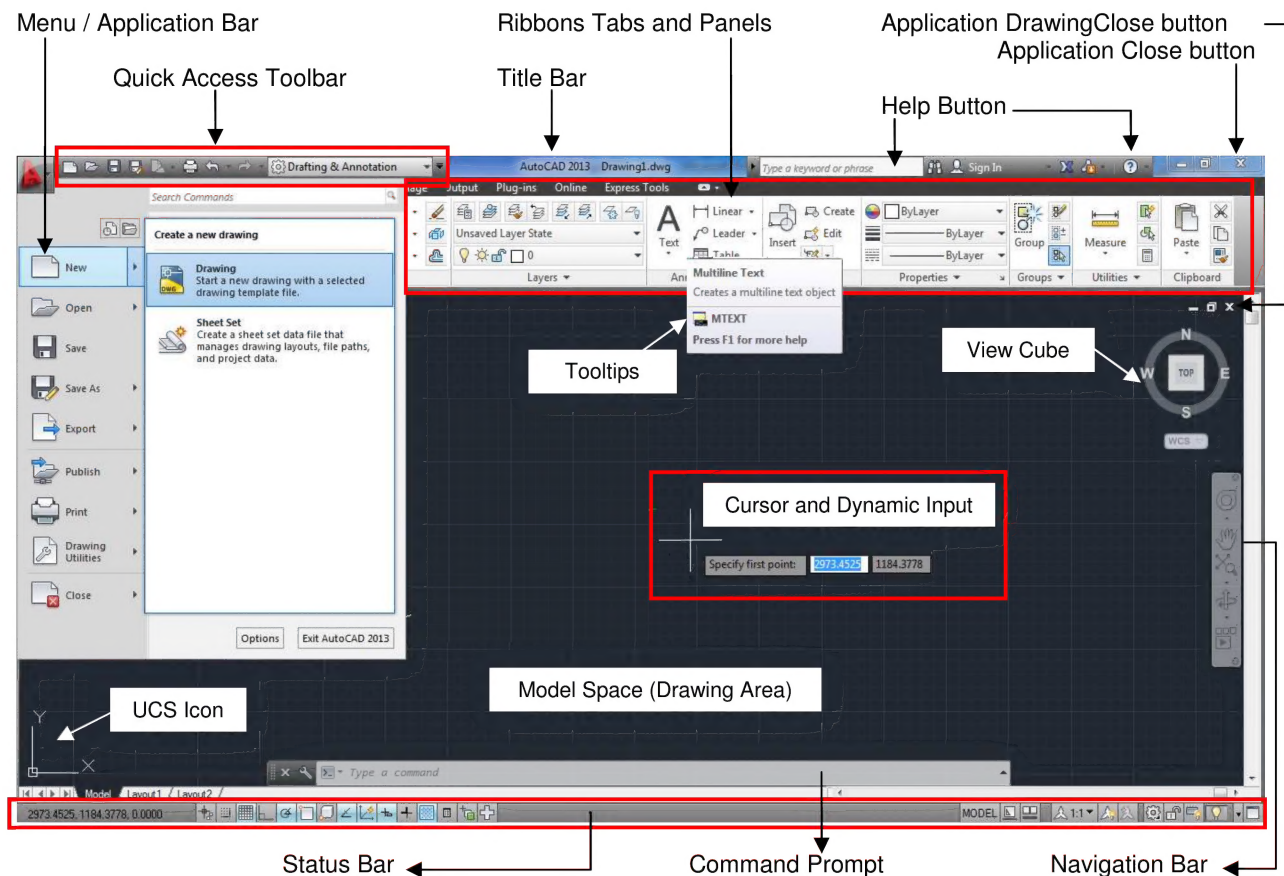
AutoCAD use Cartesian (x,y) coordinates to locate a point in the drawing where units are measured horizontally in terms of X and vertically in terms of Y. **World Coordinate System (WCS)** is the default X-Y coordinate system. If it is modified, it becomes a User Coordinate System (UCS).



**Figure 1.1:** Coordinate system and angular measurement

## 1.2 AutoCAD User Interface

Figure 1.2 illustrates the parts of the AutoCAD 2013 workspace when you launch the software. The user interface configurations may vary depending on user settings.



**Figure 1.2:** The AutoCAD 2013 **Drafting & Annotation workspace** with its various parts.

## 1.3 AutoCAD Workspaces

**Workspaces** are sets of menus, toolbars, palettes, and ribbon control panels that are grouped and organized so that you can work in a custom, task-oriented drawing environment.

There are three standard workspaces accessible:

- (i) **Drafting & Annotation:** Configured for a 2D drafting environment;
- (ii) **3D Basics/3D Modeling:** Configured for a 3D modeling environment;
- (iii) **AutoCAD Classic:** Opens a previous version of AutoCAD before 2009 user interface with pull down menus and icon toolbars.


## 1.4 Defining Positions (Entering Points in AutoCAD)

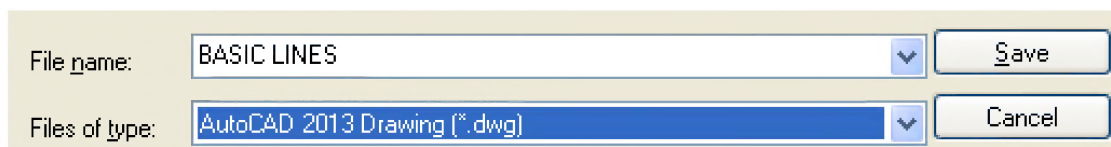
In AutoCAD, there are five methods for specifying the locations of points when we create planar geometric entities:

- (i) **Interactive method:** Use the cursor to select on the screen with **Object Snap (OSNAP)** to locate positions on existing elements in your drawing.
- (ii) **Absolute coordinates (Format: X,Y):** Type the X and Y coordinates to locate the point on the current coordinate system relative to the origin of the WCS.
- (iii) **Relative rectangular coordinates (Format: @X,Y):** Type the X and Y coordinates relative to the last point.
- (iv) **Relative polar coordinates (Format: @Distance<angle):** Type a distance and angle relative to the last point (draw a line a certain distance at a particular angle).
- (v) **Direct Distance entry technique:** Specify a second point by first moving the cursor to indicate direction and then entering a distance.

### STEP 1

#### Saving the Drawings

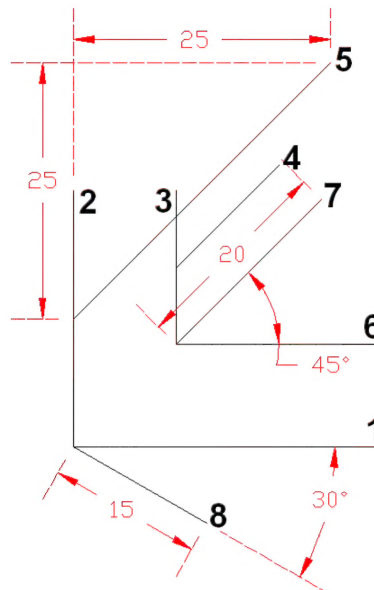
- 1) **Resize** your windows to full screen using *Maximize window*  so that it fills up the entire computer screen for larger illustrations.
- 2) Use the current file and **SAVE** your drawing file as **BASIC LINES** as in Figure 1.3. Make sure you save inside your own folder as instructed by the instructor.



**Figure 1.3:** Save as drawing (\*.dwg) file.

**STEP 2****Draw Lines Using Coordinates and Drafting Tools**

Use the **Line command** and draw the following lines in **Figure 1.4** below.



**Figure 1.4: Basic Lines**

- 1) For **Line 1**, draw a horizontal line **30mm long** that begins at the **(0,0)** point. Complete the first line by specifying the next point **(30,0)**.
  - a) Command : **LINE** or <click **Line button**,>
  - b) Specify first point: **0,0**
  - c) Specify next point or [Undo]: **30,0**
  - d) Specify next point or [Undo]: <press **Enter** to exit line command>
- 2) **Line 2** begin at the **(0,0)** point and should be **vertical, 25mm** long.
- 3) **Line 3** begin at **(10,10)** and should also be **vertical, 15mm** long.

**Note:** - To get a **straight line** you need to turn on the **ORTHO** mode on the status bar. Ortho is short for *orthogonal*, which means either vertical or horizontal.

- Try **change the position of the UCS icon**. Noorigin makes the UCS icon at lower left corner of screen. However it DO NOT change the UCS coordinates at (0,0) :

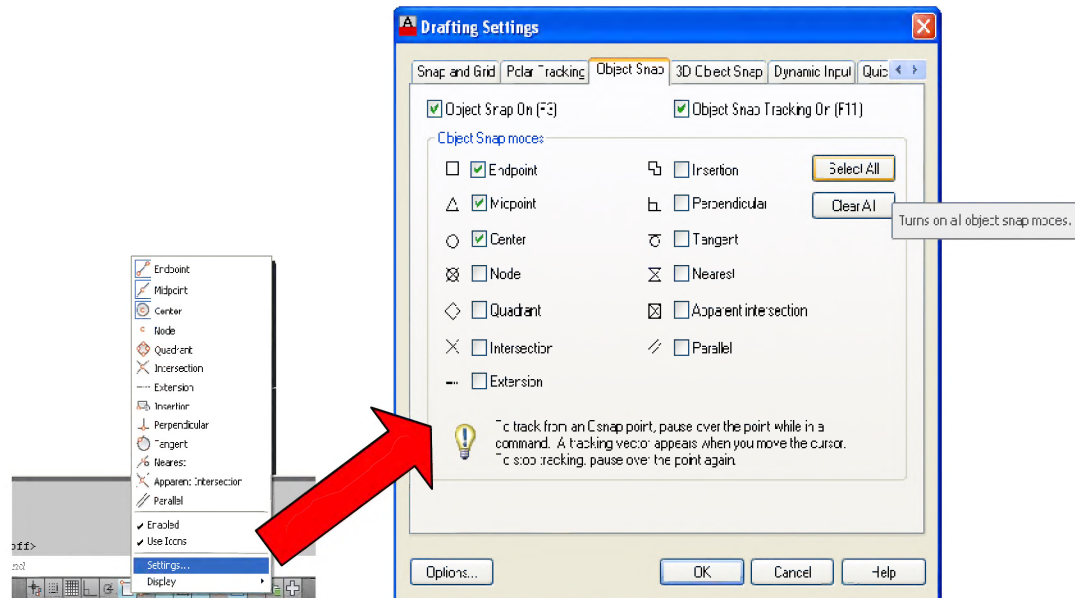
- a) Command : **UCSICON** <press enter>
- b) Enter an option [ON/OFF/All/Noorigin/ORigin/Selectable/Properties] <ON>: **N** < type **N** for Noorigin or **OR** for Origin and press enter>



**STEP 3**


The next lines in **Figure 1.4** will use a combination of these locating methods; **Relative or Polar Coordinates** and **Object Snaps**.



**Object snap (OSNAP)** are powerful AutoCAD tools that allow you to locate positions on existing elements in your drawing. You may turn all OSNAP modes as necessary.



**Figure 1.5 :** Right click on **OSNAP** for Object snap tool settings.

- 1) **Line 4** should be started from the **midpoint** of **Line 3**.
  - a) Command: **Line**
  - b) Specify first point: **<click midpoint of Line 3>**
  - c) Specify next point or [Undo]: **@10,10**
  - d) Specify next point or [Undo]: **<press Enter to exit line>**

**Note:** **Hover the cursor** over the **midpoint** of **line 3** and click the left mouse button when the cursor changes shape to .

- 2) Using the same method, start **Line 5** from the **midpoint** of **Line 2** and end it **25 along the X-axis** and **25 along the Y-axis**. Don't forget the @ symbol.
- 3) Start **Line 6** at the **bottom end** of **Line 3** (use the **Snap to Endpoint**;  or click when the cursor change to ) and make it **20mm long** and **horizontal**.

- 4) Draw **Line 7** begins at the **intersection** of Lines 3 and 6, **20mm long**, at **45 degree angle**.
  - a) Command: **Line**
  - b) Specify first point: **<click when the cursor changes shape to  $\times$ >**
  - c) Specify next point or [Undo]: **@20<45**
  - d) Specify next point or [Undo]: **<press Enter to exit line>**
- 5) Using the same method, draw **Line 8** starts at the beginning of Line 1, **15 mm long** and at a **-30 degree angle**.
- 6) Save and close your work.

**Note :** To check your lines, you can use an inquiry command called **DISTANCE**. AutoCAD will display the position, length, and angle for this line in the command prompt window. For more details on dimensioning you can refer **Tutorial 6**.

- a) Command: **DIST**
- b) Specify first point: **<click Snap to Endpoint to select one end of a line>**
- c) Specify second point: **<click Snap to Endpoint again to pick the other end>**

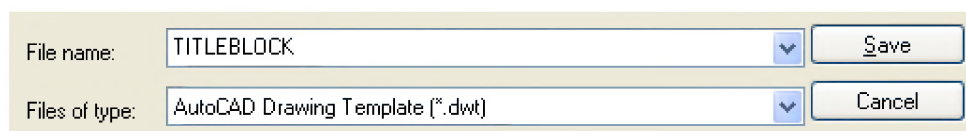
#### **STEP 4**

#### **CREATING A TITLEBLOCK**

You have to **create your own TITLEBLOCK** as shown in **Figure 1.18** at the end of this tutorial

1. This drawing template files store your settings, styles, and additional data. **YOU WILL USE THIS DRAWING TEMPLATE ON ALL DRAWINGS IN THE CLASS.**

- 1) Create a new drawing file and **Save As TITLEBLOCK** as in Figure 1.6 below **in your own folder**. Choose a *drawing template (\*.dwt)* from Files of type.

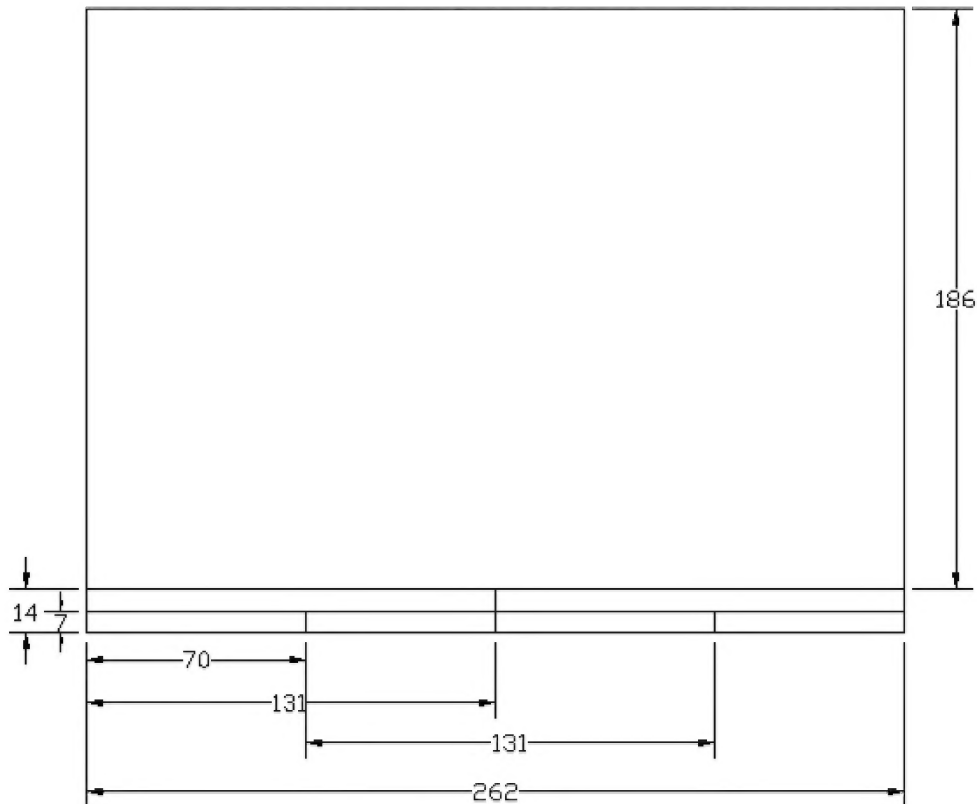


**Figure 1.6:** Save as drawing template file.

**Note:** By default, drawing template files are stored in the template folder where can accessible from the *Menu Browser* under *File>New>Select template>acad.dwt*. Just click OK when the Template Options dialog box appears.



2) The measurement of the titleblock that you have to construct is shown in **Figure 1.7** below.



**Figure 1.7:** Titleblock measurement

3) The **first line** must start at point **(0,0)** see Figure 1.8 below.




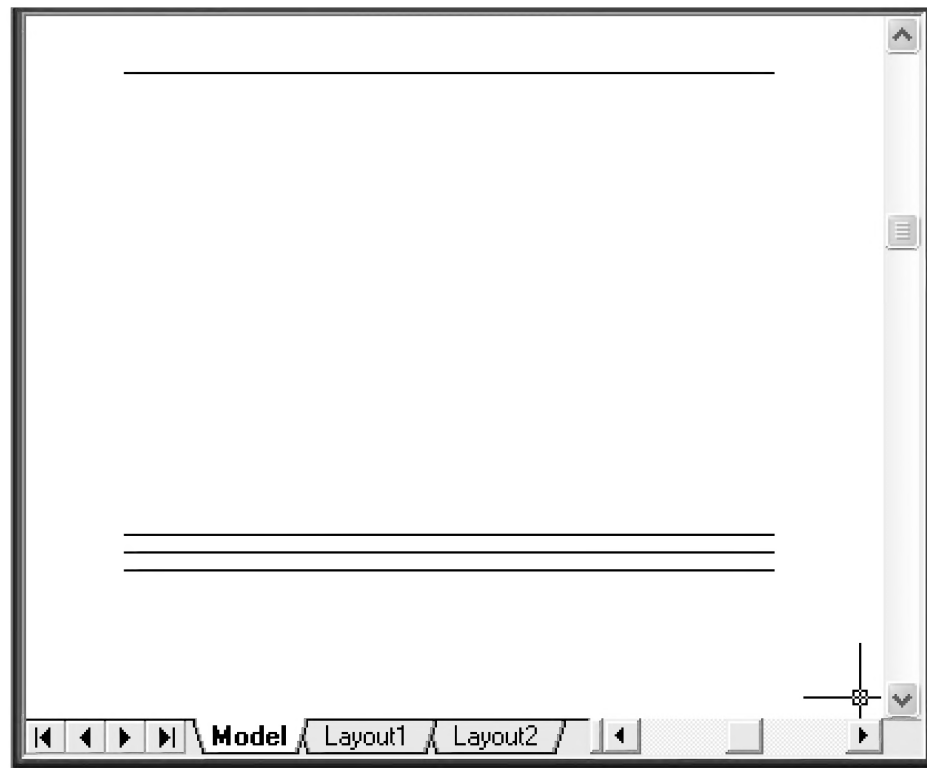
**Figure 1.8:** First line of the Titleblock

- Command : **LINE**
- Specify first point: **0,0**
- Specify next point or [Undo]: **262,0**
- Specify next point or [Undo]: **<press Enter to exit line command>**

**STEP 5**

**Offset** command copies or creates a new object parallel to the shape of a selected object. Offsetting a circle or arc creates a larger or smaller circle or arc, depending on which side you specify for the offset.

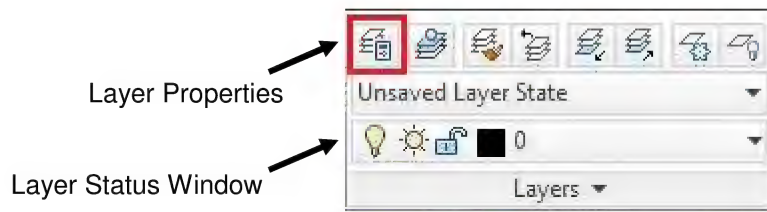
- 1) Command : **OFFSET** or <click **Offset button**,  >
  - a) Specify offset distance : **7**
  - b) Select object to offset : **<click on the first line you added >**
  - c) Specify point on side to offset: **<click anywhere in the drawing area ABOVE the line>**.
  - d) Select object to offset : **<press Enter to exit the command>**.
- 2) **OFFSET** AGAIN to add the **3<sup>rd</sup> line 7 mm above** the 2<sup>nd</sup> line and the **4<sup>th</sup> line 186 mm above** the 3<sup>rd</sup> line (See Figure 1.9).
- 3) **Zoom All** to see all lines. **FINISH ALL THE LINE IN FIGURE 1.7** using the command and drafting tools that you learn earlier.





**Figure 1.9: Using Offset**

**STEP 6**

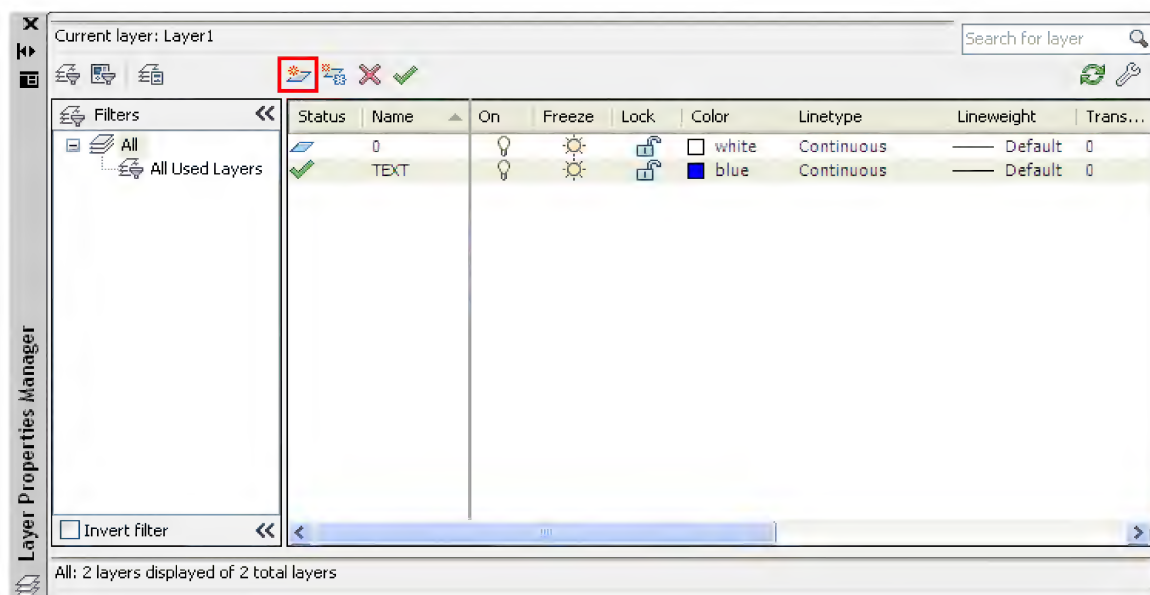
Before adding text, you must create a new **LAYER**. By default, every new drawing will automatically create a 0 layer. You cannot rename or delete the default layer.



**Figure 1.10:** Current layer shown in Layers Panel

- 1) Click the **Layer Properties** button (Figure 1.10) on Layers Panel under Home Tab.
- 2) Click on **New Layer** button,  (the little yellow explosion just above the Status column) on the **Layer Properties Manager** (Figure 1.11) to create a new layer.
  - a) Type the new layer name as **Text** in the name slot.
  - b) Change the color to **blue** and click **OK**.
  - c) Click the **Set Current Button**,  to accept the new layer settings.
  - d) Close the Layer Properties Manager dialogue box once finish.

**Note:** The Layer Status Window drop-down list on the Layers Panel now displays the **Text Layer** as the current layer. Everything drawn within this layer is **blue** in color.




**Figure 1.11:** Layer Properties Manager dialogue box.

**STEP 7**

Add **TEXT** to the Titleblock.

**A**

- 1) Command : **DTEXT** or click  in *Home Tab > Annotation Panel > Single Text*
  - a) Specify start point of text or [Justify/Style]: **2,2**
  - b) Specify height: **3**
  - c) Specify Rotation angle of text : **0**
  - d) <Type the text> **SUBJECT:**
  - e) <Enter until you exit the text command>
- 2) Repeat the DTEXT command again and enter the text for the following position.
  - a) For position **2,9**, same height , same rotation, type in **DRAWN BY: <YOUR NAME>**
  - b) For position **72,2** , same height , same rotation, type in **TITLE: TUTORIAL 1**
  - c) For position **133,9** , same height, same rotation, type: **COURSE: <YOUR COURSE>**
  - d) For position **133,2** , same height, same rotation, type: **SCALE: 1:1**
  - e) For position **202,2** , type **DATE: <TODAY'S DATE>**
- 3) Change back to **0 layer**. Save your file. Your drawing should look like **Figure 1.12** below.

DRAWN BY :		COURSE:	
SUBJECT:	TITLE:	SCALE:	DATE:

**Figure 1.12:** Text in the Titleblock.

**STEP 8**

**Block** command allows you to select part or all of the drawing, group it, and reuse it over and over. Once it is a block, you can use **Insert** to place as many copies of a block you need.

- 1) Make sure:
  - (i) The **Layer Status Window** is in **0 layer**.
  - (ii) The **Insertion Scale** in **Millimeters** (go to *Menu browser > Drawing Utilities > Units*)
  - (iii) Zoom All if necessary to see the entire Titleblock.
- 2) Type **BLOCK** or go to *Home Tab > Block Panel > Create*. A Block Definition dialogue box as in Figure 1.13 below appears.

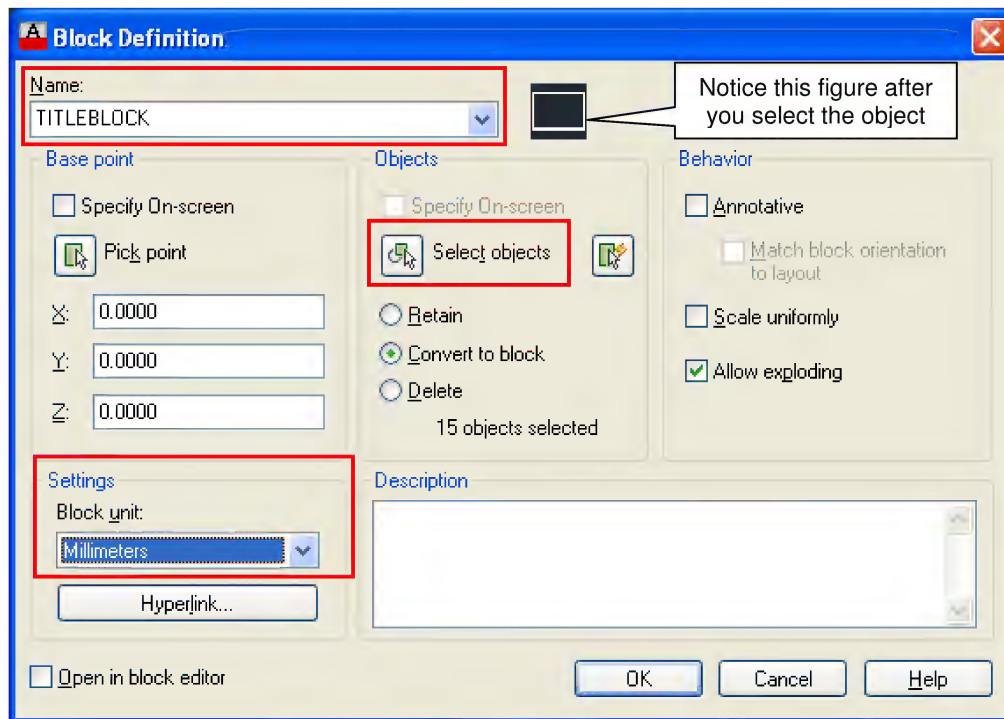


Figure 1.13: Block Definition dialogue box.

- Type **TITLEBLOCK** on Name:
- Make sure the Block Unit Settings in **Millimeters**.
- Under the heading **Objects**, click **Select Objects** button. AutoCAD will return to the drawing (model space). **Select the entire Titleblock** then **press Enter**.
- Under **Basepoint** heading, select the **absolute coordinate (0,0,0)**. In this case, just click **OK** because the X,Y and Z points is 0.000. A **Basepoint** is the position on the block that AutoCAD will use when inserting it into the **Layout or PAPER Space**.

## STEP 9

### Layout and Model Tabs

- By default at the bottom left of AutoCAD screen, there are three tabs, labeled **Model**, **Layout1** and **Layout2**. If you can't see it, **right-click** at either **model/layout icon** on the status bar as shown in Figure 1.14.

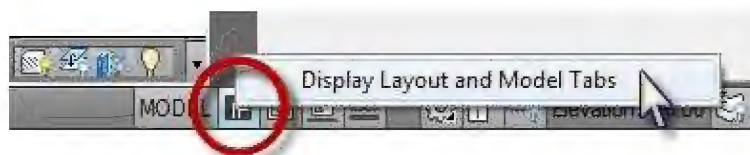



Figure 1.14: Display Layout and Model Tabs.



- 2) Click at the **Layout1** tab (  ). **Layout** is a paper space for printing. A *dashed box* represents the printable area and a continuous **box inside** is a **Viewport** where it show whatever drawings in the model space (see Figure 1.15).

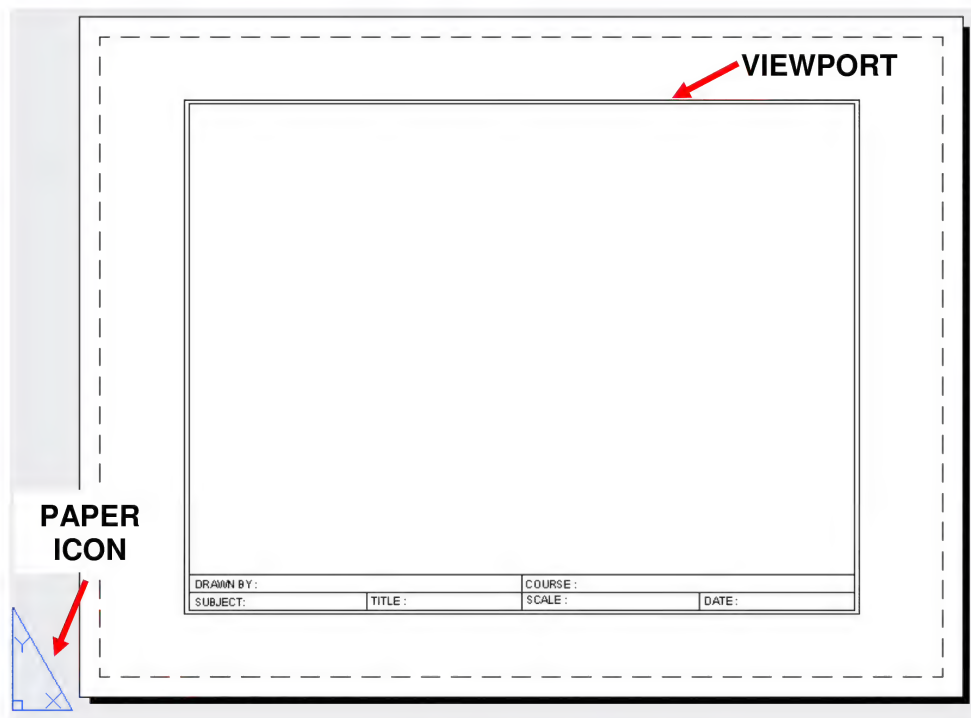

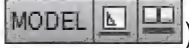


Figure 1.15: Layout 1

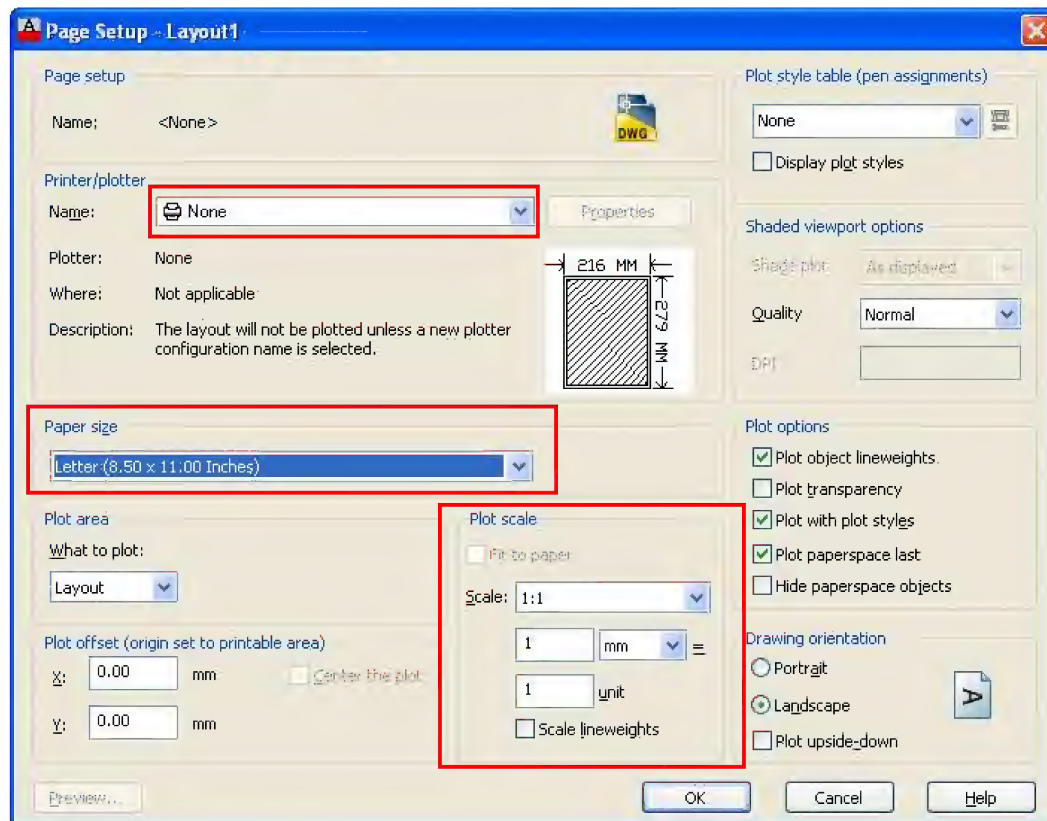
- 3) **Select the Viewport** and **ERASE** or **DELETE** it to make sure you have an empty paper space before inserting your Titleblock.

**Note:** - Make sure you are in **PAPER side of the Paper Space** before you erase the viewport to prevent from losing your drawing in the model. Notice the PAPER icon (right-angle triangle) and PAPER icon (  ) shown on the status bar.

- If you accidentally in the MODEL side of the Paper Space, double-click at any area outside the Viewport or click at the MODEL icon (  ) on status bar.

**STEP 10****PAGE SETUP IN LAYOUT**

- 1) Go to **Layout Tab**, and then click **Page Setup on Layout Panel**. A dialogue box of Page Setup Manager appears. **Choose Layout1** (current layout) and click **Modify** button to change the page settings. A Page Setup dialogue box appears as shown in Figure 1.16.



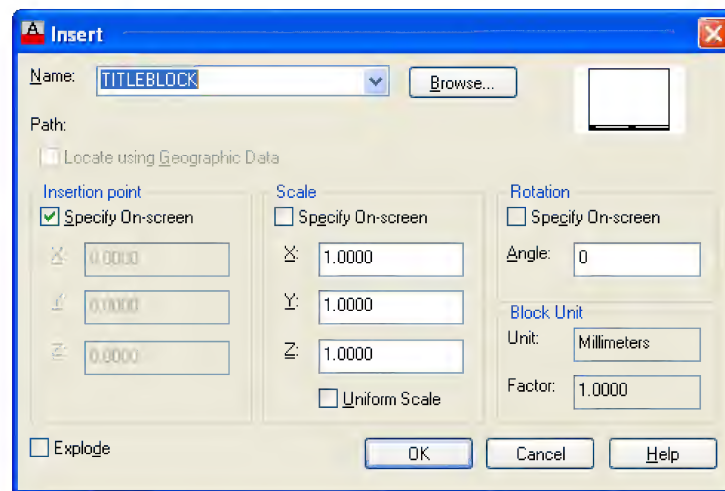
**Figure 1.16:** Page Setup dialogue box.

- 2) Make sure your settings are the same as in **Figure 1.16** above. Click **OK**.

**STEP 11****INSERT TITLEBLOCK**

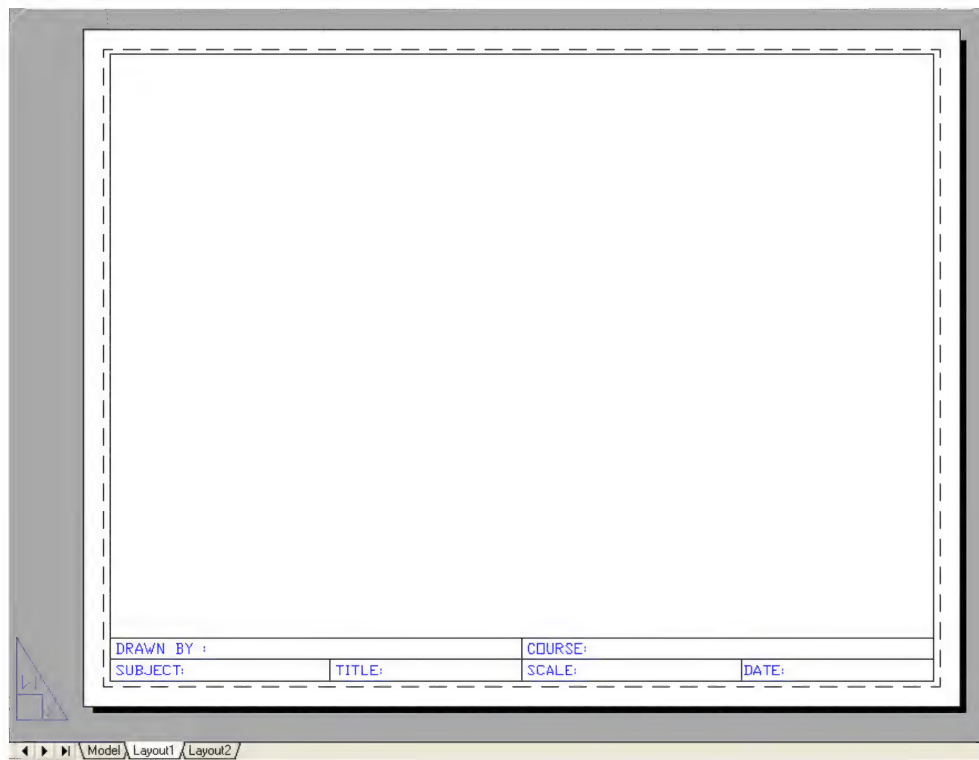
- 1) Type **INSERT** or click Insert on the Block Panel in Home Tab. Insert dialogue box will appear as shown in **Figure 1.17**. Click **OK**.

**Note:** The configuration in the Insert Dialogue Box is automatic and you can't edit it. If you get different configuration, you need to check **STEP 8** again.



**Figure 1.17:** Insert dialogue box.

- 2) You will return to the **Layout 1** screen. The titleblock will appear in the Layout1 screen, but can be moved by dragging the cursor.
- 3) Place it about center (refer **Figure 1.18**) , **save and close** your drawing.



**Figure 1.18 :** A finish Titleblock as a drawing template in Layout1.



**Important Note! :** This TITLEBLOCK.dwt will be as your template file to start from.



## TUTORIAL 2: 2D Geometry

### OBJECTIVES:

After completing this tutorial, you should be able to:

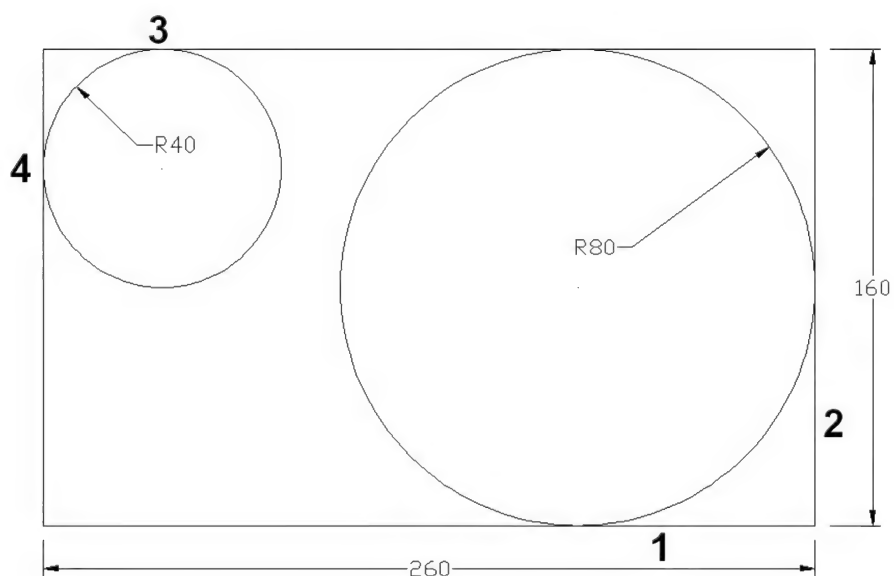
- 1) Create a two-dimensional drawing using the LINE, CIRCLE and POLYGON.
  - 2) Use the editing tools such as ROTATE, TRIM and EXTEND commands;
  - 3) Create centre lines;
  - 4) Insert a drawing into the Titleblock; and
  - 5) Print a drawing to scale.
- 

### STEP 1

- 1) Open a new drawing template from **File> New > Drawing**.
- 2) Locate your **Titleblock.dwt template** from the “Select Template” dialog box.
- 3) **Save As** the file as **Tutorial2** inside your own folder.
- 4) Change back to the **Model tab**, erase the Titleblock but make sure the Titleblock still remain in Layout1 after you erase it.


### STEP 2

- 1) In the **model space**, draw a rectangle in **Figure 2.1 below**, using a series of **straight lines** and **Object Snaps** with the given measurement.



**Figure 2.1:** Adding circle between lines.

2) Draw **the Circle Between Lines** Using **Tan Tan Radius (ttr)**

- a) Command : **CIRCLE** or <click the Circle button;  >
- b) Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **ttr** or **t**
- c) Specify point on object for first tangent of circle: **<click the first line>**
- d) Specify point on object for second tangent of circle: **<click the second line>**
- e) Specify radius of circle: **80**


3) Command : **CIRCLE**

- a) Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **ttr**
- b) Specify point on object for first tangent of circle: **<click the third line>**
- c) Specify point on object for second tangent of circle: **<click the fourth line>**
- d) Specify radius of circle: **40**

4) Your drawing should be like in **Figure 2.1**.


**STEP 3**

The TRIM command allows you to shorten an entity to an intersection or remove a section of an entity between two intersections. TRIM button is in *Home tab > Modify panel > Trim*.

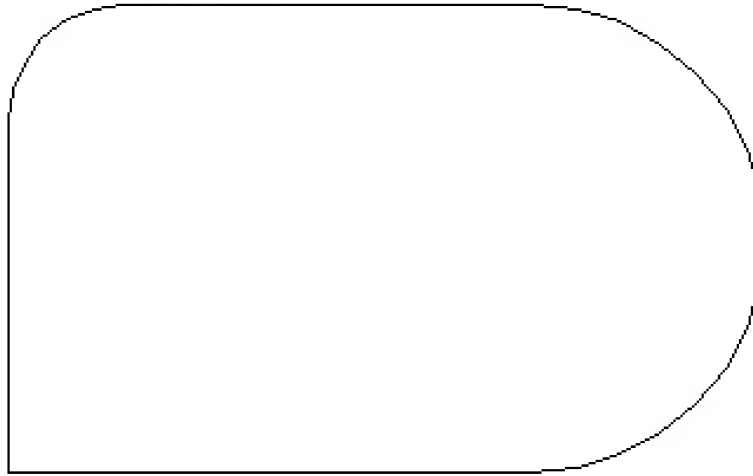
- 1) Command: **TRIM** or <click the Trim button;  >
  - a) Select objects or <select all>: **All**
  - b) Select objects: **<press Enter>**
  - c) Select object to trim or shift-select to extend : **<Click on the line to trim>**
  - d) **Press Enter once finish.**

**Note:** You can use **Undo** to replace the line if you accidentally remove the wrong one.

Remove the excess vertical line (touching the large circle) with the **Erase** command or just use the delete button on the keyboard.

- 2) Command: **ERASE** or <click the Erase button;  >.
  - a) Select objects: **<Click on the line>**
  - b) Select objects: **<Press Enter>**
  - c) Examine Figure 2.2 and save.

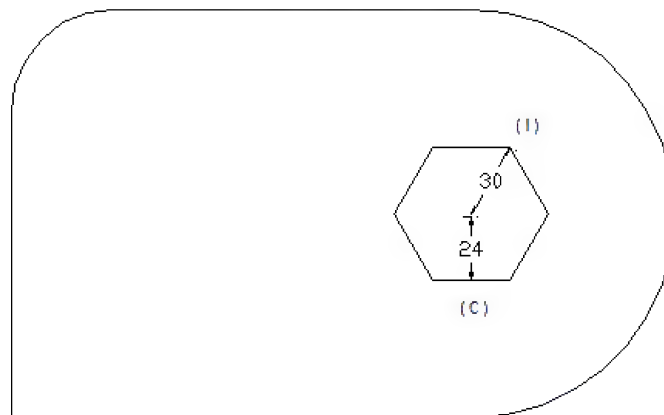




**Figure 2.2:** TRIM command removes part of object based on cutting edge.


#### **STEP 4**


The next step is to add a **hexagon** to the drawing and modify it.



**Figure 2.3**

**Note:** - (i) represent **inscribed** (for across the corners): Specifying the radius when you know the distance between the center of the polygon and the endpoint of each side.  
- (c) represent **circumscribed** (for across the flats): Specifying the radius when you know the distance between the center of the polygon and the midpoint of each side.

- 1) Command: **POLYGON** or < click the Polygon button;  in Home tab > Draw panel >
  - a) Enter number of sides: **6**

- b) Specify center of polygon or [Edge]: **< hover the cursor at the circle until  symbol (Snap to center) appears indicating the center of the circle and click >**
- c) Enter an option [Inscribed in circle/Circumscribed about circle]: **I** *<see Figure 2.4>*
- d) Specify radius of circle: **30**

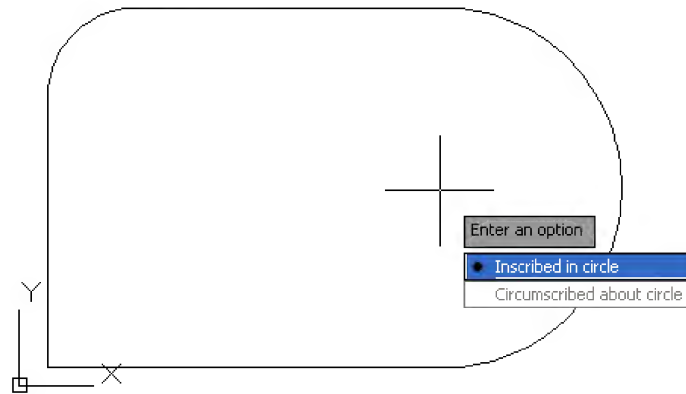



Figure 2.4

The hexagon you just added has a corner at **0 degrees on the XY plane**. This is a hexagon's default orientation in AutoCAD. You need to **rotate the hexagon** so that this corner is at a **90 degree angle (aligned with the Y-axis)**.

- 2) Command: **ROTATE** or *<click the Rotate button;  in Home tab > Modify panel >*
  - a) Select objects: **<click on the Hexagon you wish to rotate>**
  - b) Select objects: **<press Enter>**
  - c) Specify base point: **< Use the Snap to Center to specify the base point of rotation>**
  - d) Specify rotation angle or [Copy/Reference] *<0>*: **90**

**Note :** The object will rotate the degrees indicated. The angle can be a positive or negative number. See Figure 2.5.

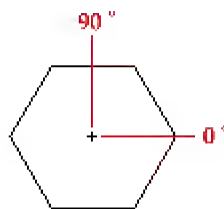




Figure 2.5

A polygon is a group or single entity in AutoCAD. To modify a polygon, you must **explode** it first. It is not possible to remove part of it otherwise.

- 3) Command: **EXPLODE** or <click the Explode button;  in Home tab > Modify panel >.
  - a) Select objects:<**Click on the Hexagon**>
  - b) Select objects: <**Press Enter**>
- 4) With the hexagon exploded, use the **Erase button**;  to remove the two top lines of the hexagon. Look at Figure 2.6 to be sure which lines to remove. **Save** your changes.

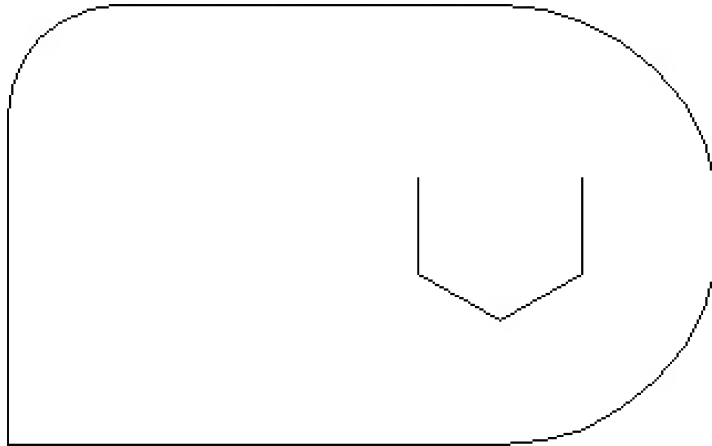



Figure 2.6

### **STEP 5**

The next command you will use is the **Extend** command. This command will allow you to extend the vertical sides of the hexagon so they touch the top of the drawing.

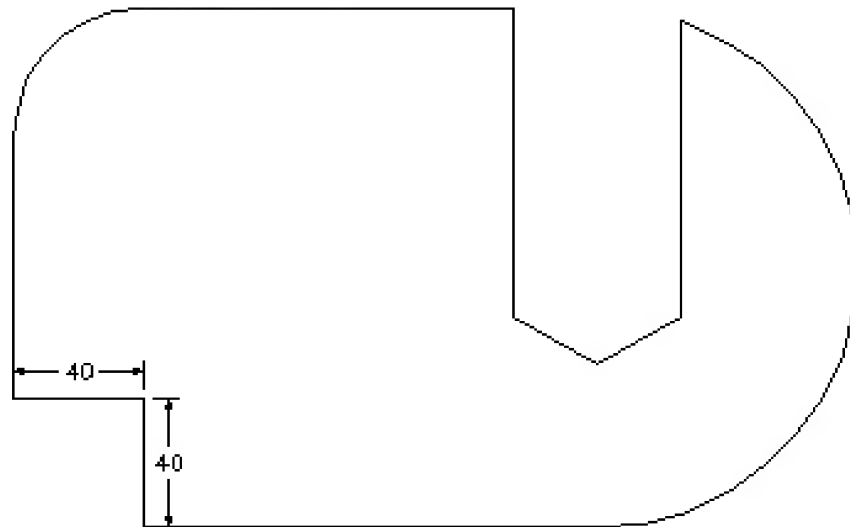
- 1) Command: **EXTEND** or <click the Extend button,  >.
- 2) Select objects or <select all>: **all** or <Click on the item(s) that you wish to use as a boundary>.

**Note :** The boundary is the object you want to extend an element to (you can select more than one boundary edge so that multiple extensions can be made at one time).


- 3) Select objects:<**press Enter** or click the right mouse button>
- 4) Select object to extend or shift-select to trim:<**Click on the end of the line you want to extend (on the side that should be extended)**>
- 5) If you have other lines to extend, you can continue selecting them. Press Enter to exit the command.

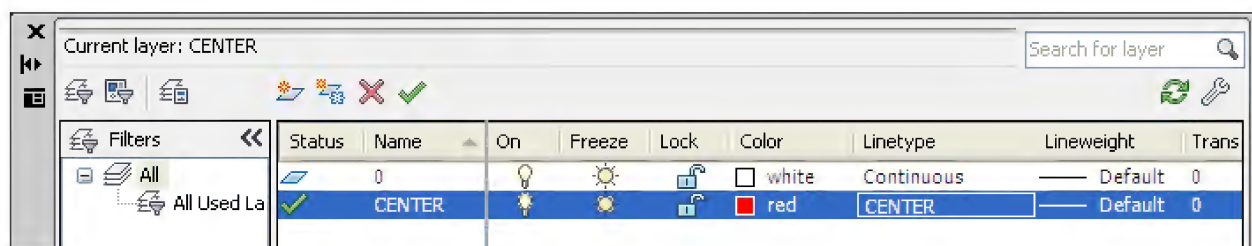
**STEP 6**

- 1) Use the **Trim command** to remove the part of the horizontal line and the arc, between the lines you extended as in Figure 2.7 below.
- 2) **Add the lines** at the lower left corner of the figure. Use the **Trim command** to remove the unwanted part of these lines.

**Figure 2.7****STEP 7**

Now that the visible lines of the drawing are complete, you need to **add center lines**.

- 1) Click the **Layer Properties button**;  on Layers Panel and **create a new layer** named **Center**.
- 2) Click on **Color** and select your favourite color.
- 3) Click on **Linetype** and choose **Center** ,if there is no center line, you have to click **Load** and find the Center line. Click **OK**.
- 4) Make sure you set the **Center Layer as the current layer** as in Figure 2.8.

**Figure 2.8**

- 5) To add center lines to the arcs in your drawing:
- First you will **add two circles** as in Figure 2.9.
  - Use the **Circle** command and identify their centers with the **Center Osnap** so they are **concentric** with the **large and small arcs**.
  - The circle that is concentric with the large arc should have a radius of **90mm**, and the circle that is concentric with the small arc should have a radius of **50mm**.

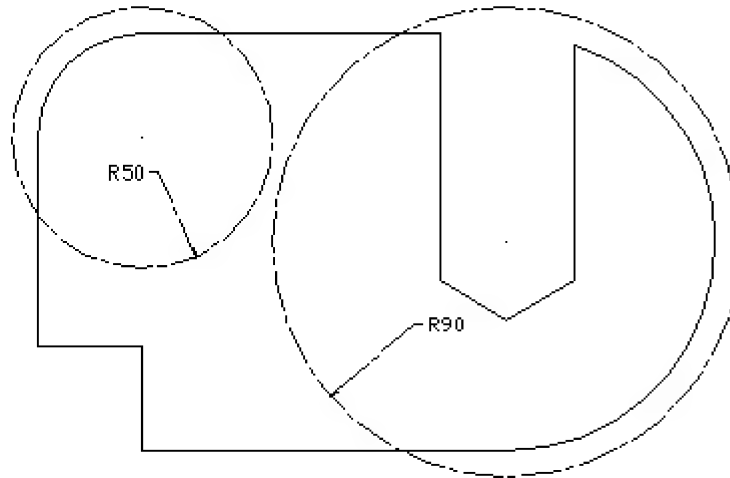




Figure 2.9

- 6) Once the circles are placed, draw horizontal and vertical center lines across each arc using the **Snap to Quadrant**;  (see Figure 2.10 and Figure 2.11).
- Command : **LINE**
  - LINE Specify first point:< **hover the cursor to one side of the circle and click when**  **symbol (Snap to Quadrant) appears indicating the quadrant of the circle**>
  - Specify next point or [Undo]:< **click the other quadrant**>
  - Specify next point or [Undo]:<**press Enter to exit command**>

**Note:** Notice that the **cursor changes** to a "diamond" shape when using this and "snaps" to the **appropriate position on the edge of the circle**.



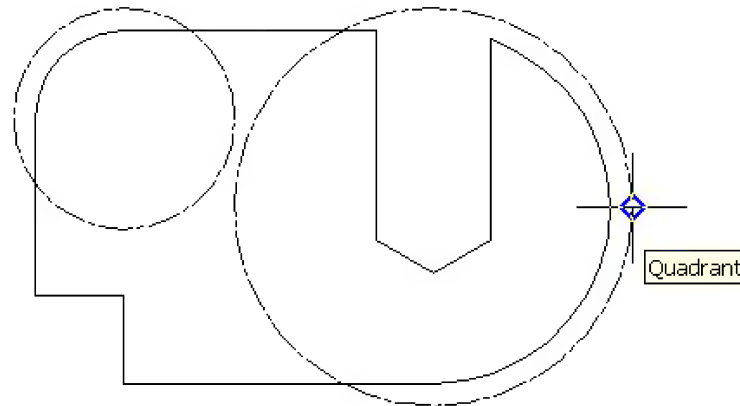


Figure 2.10

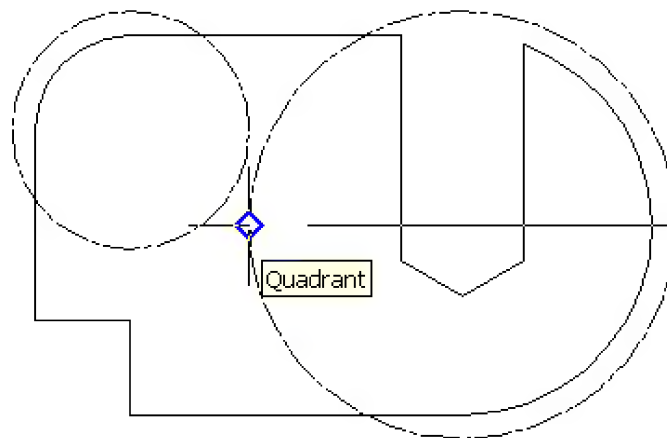


Figure 2.11

- 7) Repeat the same steps to create **horizontal** and **vertical center lines** for both circles. See **Figure 2.12 for the line positions.**
- 8) Finally, **Erase** the **circles** you added for the construction of the lines.

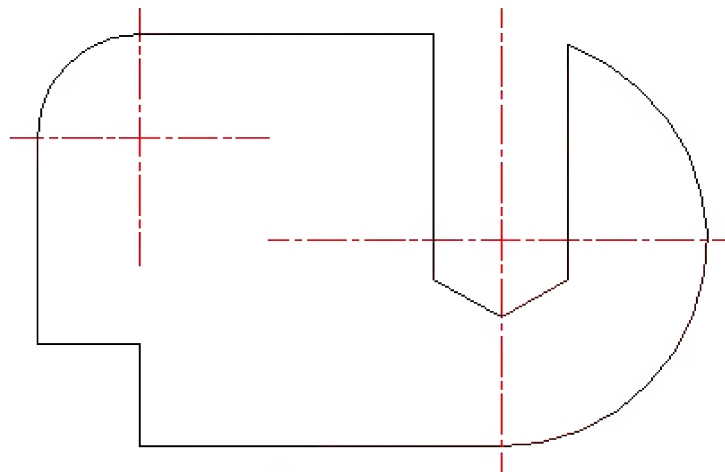
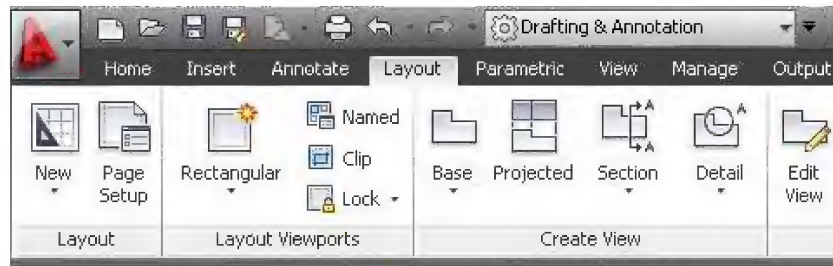


Figure 2.12

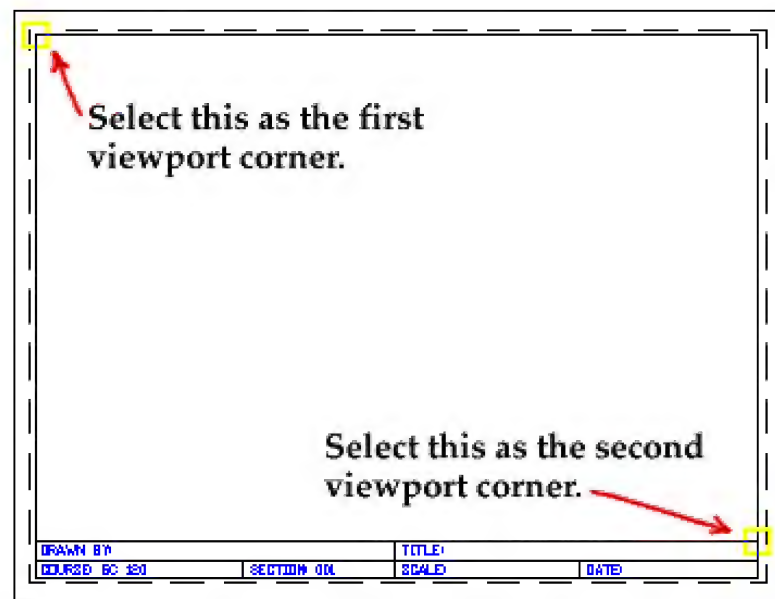
**STEP 8****VIEWPORT**

Finally, it is time to **insert the drawing in the Titleblock on Layout1** that you created in the first tutorial. Make sure you are in the **Paper side of the Paper Space**.



**Figure 2.13:** Layout Viewports Panel under Layout Tab

- 1) Select the **Rectangular** button on Layout Viewports Panel
- 2) Use the **Endpoint or Intersection Osnap** to select the first and second corner of the titleblock for the viewport frame (see Figure 2.14).
- 3) The drawing should now inside the titleblock.



**Figure 2.14**

- 4) For this drawing, you need to scale the drawing to **1:2 (half size)** before you print.
- 5) To scale the drawing, you must first **select the viewport frame** (Figure 2.15).

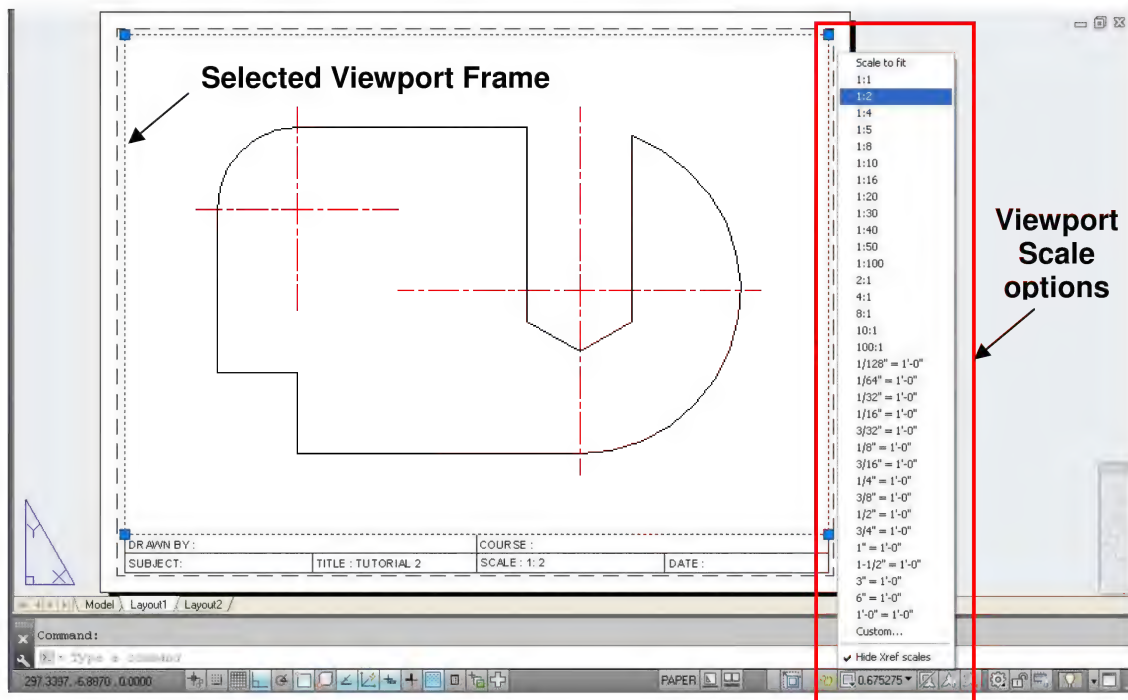


Figure 2.15

- 6) With the viewport selected, you can click on the **Viewport scale arrow** at the right of the window containing the current size of the drawing and choose the **1:2** scale.
- 7) **Save and close your drawing.**

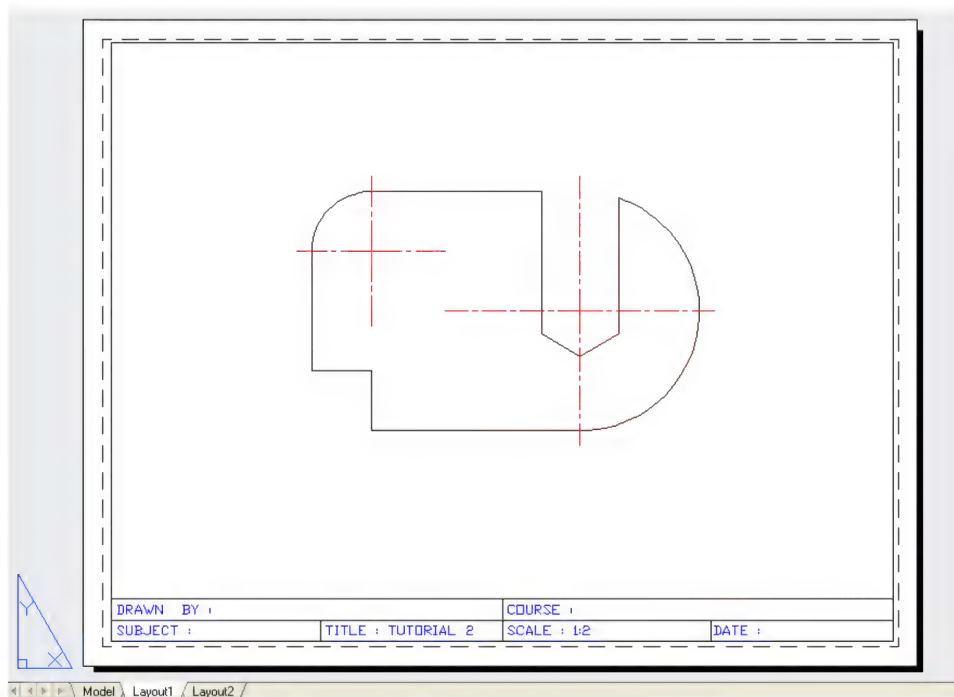


Figure 2.16: A finish Tutorial 2 drawing.



## TUTORIAL 3: Create 3D from 2D

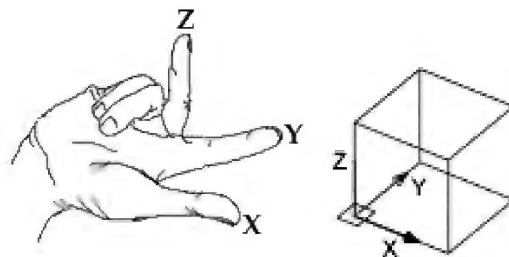
### OBJECTIVES:

After completing this tutorial you should be able to:

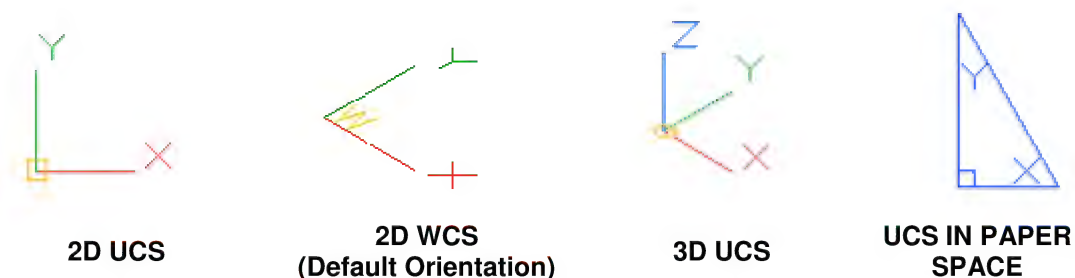
- 1) Understand and familiarize with 3D environment;
- 2) Create a 3D forms from 2D shapes;
- 3) Create solid models using EXTRUDE, REVOLVE and PEDIT commands;
- 4) Insert a model into the titleblock, scale it, and print it.

### 3.1 Introduction

**3-Dimensional** (3D) is a way of displaying real-world object in a more natural way by adding depth to the height and width. This system uses **the X Y and Z axes**. 3D is all about the third Z coordinate. In 2D, we only care for the X and Y axes, but never used the Z axis. The Z axis is obtained directly from the X and Y axes by the **right-hand rule**: if we rotate the right hand from the X axis to the Y axis, the thumb indicates the positive Z direction (see Figure 3.1).



**Figure 3.1:** World Coordinate System (WCS).



**Figure 3.2 :** AutoCAD displays the UCS icon differently for 2D, 3D and Paper Space environments.

### 3.2 Creating 3D Forms from 2D Shapes

Working in 3D usually involves the use of solid objects. The 3D solid primitives are great for creating some basic shapes, but in many situations, you will want to **create a 3D form from a more complex shape**. Fortunately, you can **extrude or revolve** from a closed 2D shape to create 3D solid using tools in **3D Modelling Workspace**.

#### STEP 1

- 1) Open a new drawing template from **File> New > Drawing**.
- 2) Locate your **Titleblock.dwt template** from the “Select Template” dialog box.
- 3) **Save As** the file as **Tutorial 3** inside your own folder.
- 4) Click on the **Model tab**, to move to **MODEL Space (drawing area)**. Your model must be created in 3D Modeling workspace.
- 5) **Figure 3.3** provides a preliminary view of the model you will construct, with the subparts identified.

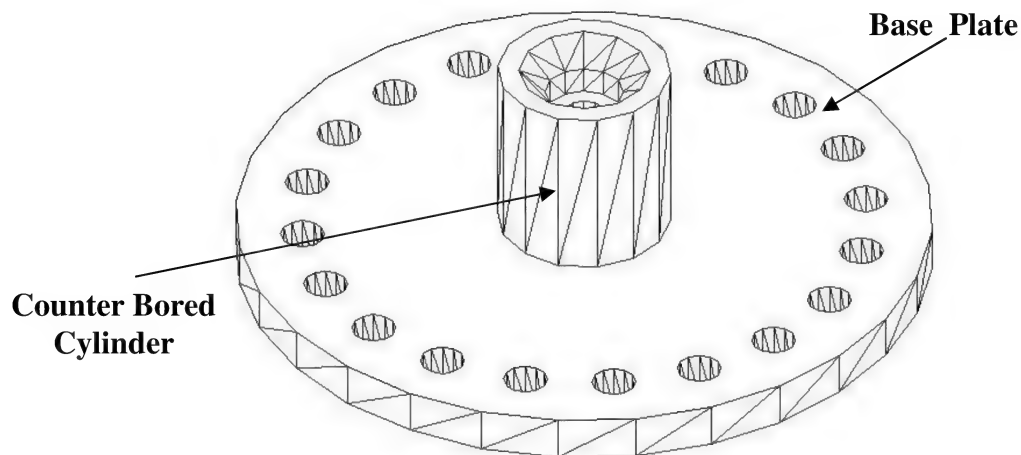



Figure 3.3

#### STEP 2

##### CREATING THE BASE PLATE

To construct this model, you will create a series of **two-dimensional elements** that can be **converted to a solid** to form the first part of the model, the **Base Plate**.



Change the view by using the **vp** command (**X Axis = 270°**, **XY Plane = 90°**) or click **Top View** button;  from View Panel.

The first element you need for the **Base Plate** is a **circle**. Draw the circle with centre at coordinate 0,0 and radius **80**. Circle can be found at Draw Panel under the Home Tab.

To the large circle, add a **5 mm radius** circle that is **vertically aligned** with the center of the larger circle and **65 mm** above it. **See Figure 3.4.**

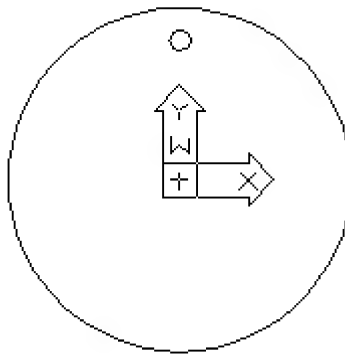


Figure 3.4

### STEP 3

With the small circle in place, you will use a new command, called **Array**, to make multiple copies of the small circle. There are **two Array commands** in AutoCAD, one for **2D elements** and one for **Solids**. Each **Array** command is capable of producing three types of arrays, **Rectangular**, **Polar** and **Path**.

To access this command, type **Array** at the command prompt or select Polar Array Button;

 **Array** on the Modify Panel under Home Tab as in **Figure 3.5.**

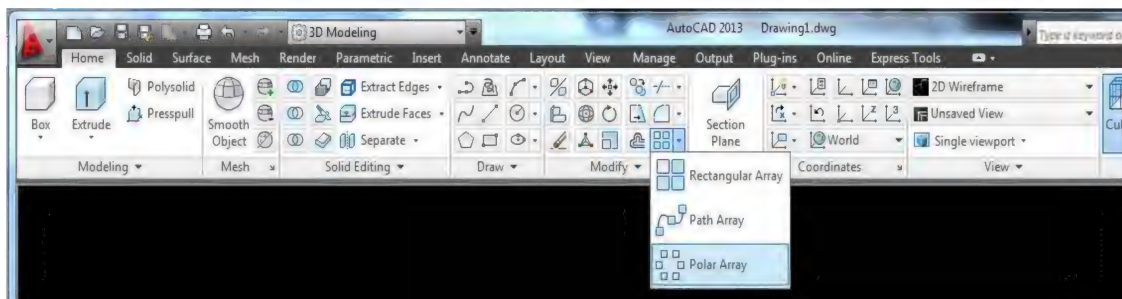


Figure 3.5

- 1) Command: Array
  - a) Select Objects:<**select the object to array and press Enter**>
  - b) Enter Array Type:<**Polar**>
  - c) Specify center point of Array:<**Click the center of circle with radius 80**>

Note: By default, 6 objects distribute around the center point evenly.

- 2) Notice the **Additional Array ribbon** will display the option for modification. Modify the details on the ribbon to suit your task. Refer Figure 3.6. The completed array will look like Figure 3.7.

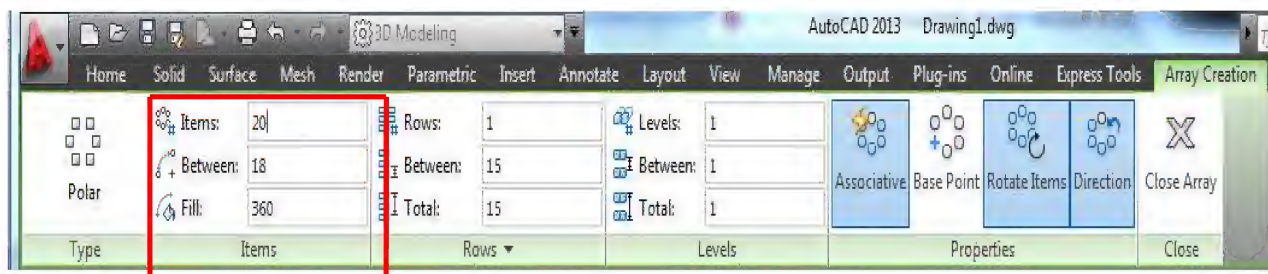


Figure 3.6

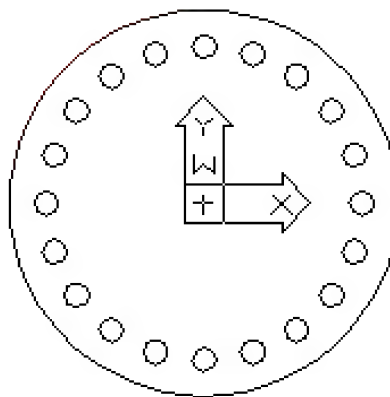



Figure 3.7

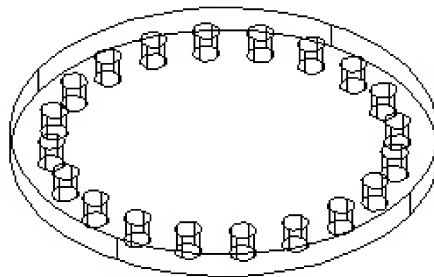
- 3) Now, type **explode** at the prompt command. Select all the small circle and press Enter. The EXPLODE command allows you to change a grouped item into its individual elements so it can be edited.

#### **STEP 4**

Next you will use the **Extrude** command to **convert the circles to solid cylinders**. Extrude all the elements you have created so they have a positive **thickness** of **10mm** and are **NOT tapered**.

- a) Command: **EXTRUDE** or <click the Extrude button;  Extrude >
- b) Select objects to extrude: **ALL**
- c) Select objects to extrude: **<Press Enter>**
- d) Specify height of extrusion or [Direction/Path/Taper angle]: **10**

To see the extrusion, you need to change your **view point**. Use the **Vpoint** command and **ROTATE** your view to **300** degrees **IN the XY Plane** and **35** degrees **FROM the XY Plane**. The image on the screen should now match the one in **Figure 3.8**.




**Figure 3.8**

### **STEP 5**

After using extrude command you can see that all objects have been extruded to the height that you have set. However, the small cylinders on the base plate is actually holes on it. You can cut them out from the plate using Subtract command.

Use the Subtract command to **remove the small cylinder** from the base plate. Look at the Figure 3.8 to see how the figure should now look.


- a) Command: **Subtract** or < click subtract button;  at **Solid Editing Panel under Home Tab**>.
- b) Select objects:< Click on the base plate, press Enter>
- c) Select objects:< Click all the small cylinder, press Enter>

When you press **Enter** they will removed.

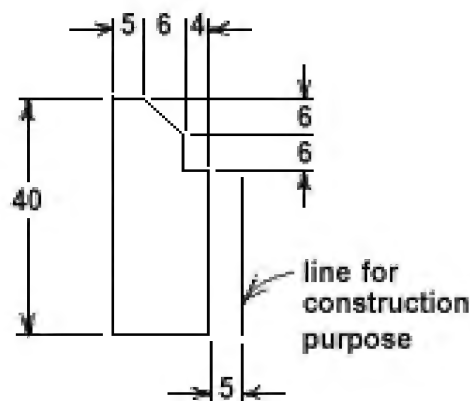
Again, the **model will not look any different** than the one in Figure 3.12. To check your model, use **Visual Style Panel** under View Tab, select Conceptual to view your model . The figure should have a holes after do subtract and completed drawing must same as Figure 3.13.

**STEP 6****CREATING THE COUNTER BORED CYLINDER**

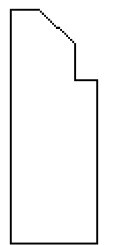
For the last part of the model, you will create a cylinder with a chamfered counter bored hole by revolving a **2D Polyline 360 degrees** to form a solid. The **Revolve** command can be used to create a circular or arched shape from any set of closed **polylines**.

Change the view by using the **vp** command (**X Axis = 270°**, **XY Plane = 0°**) or click **Front View** button;  from View toolbar. You should now be looking **directly** at the **FRONT** surfaces of the Baseplate.

Look at **Figure 3.9**, which shows a diagram of the 2D figure you will create and provides the dimensions to create it. Again, construct this figure in blank space above the baseplate on AutoCAD's drawing area. It will be moved into place after you convert it to a solid.

**Figure 3.9**

Using the diagram in Figure 3.9 create this structure. See Figure 3.10 for a view of the 2D figure when you are done.

**Figure 3.10**


Notice that you were asked to add a line to the right side of the "closed" figure, which was **5mm** from it. This is a **construction line** you will use as the axis of rotation when you rotate the figure. This will create an opening through the Chamfered-Counter bored Cylinder.

### **STEP 7**

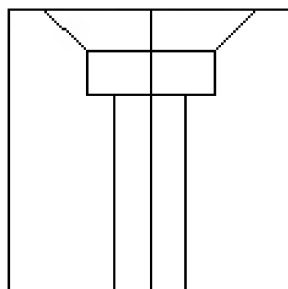
Once the figure is complete, convert it to a "closed" Polyline path with the **Pedit** command. **Do not** include the construction line.

- 1) Command: **PEDIT**
- 2) Select polyline or [Multiple]: **<Select one of the line>**
- 3) Do you want to turn it into one? <Y> **< Type Y for yes>**
- 4) Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Undo]: **Join**
- 5) Select objects: **<Select all of the line except the construction line>**
- 6) Press Enter to end the command. Make sure that all the lines are connected together.

### **STEP 8**

- 1) Command: **REVOLVE** or *<select the revolve button;*  **Revolve** *>*
- 2) Select objects to revolve: **<Select the figure you just converted to Polylines>**
- 3) Specify axis start point or define axis by [Object/X/Y/Z] <Object>: **< Use an Osnap to select one end of the construction line that is 5mm away from the structure >**
- 4) Specify axis endpoint: **< Use an Osnap to select the opposite end of the same construction line>**
- 5) Specify angle of revolution or [STart angle] <360>: **360**

Your figure should now look like a cylinder with a counter bored hole and a chamfered edged as it would appear from a FRONT view (see Figure 3.11).



**Figure 3.11**

To see the Counter bored Cylinder as a 3D object, change your **viewpoint** to **300 degrees IN the XY plane** and **35 degrees FROM the XY plane** (see Figure 3.12). Now, you can delete the construction line when everything is done.

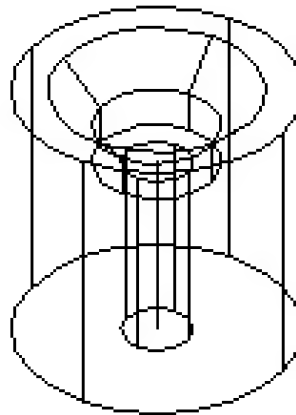


Figure 3.12

#### **STEP 9**

Use the **Move** command and select the **Center** of the **BOTTOM** of the **Counter bored Cylinder** as the **basepoint for this move**. Use **Union** command to complete the structure as in Figure 3.13.

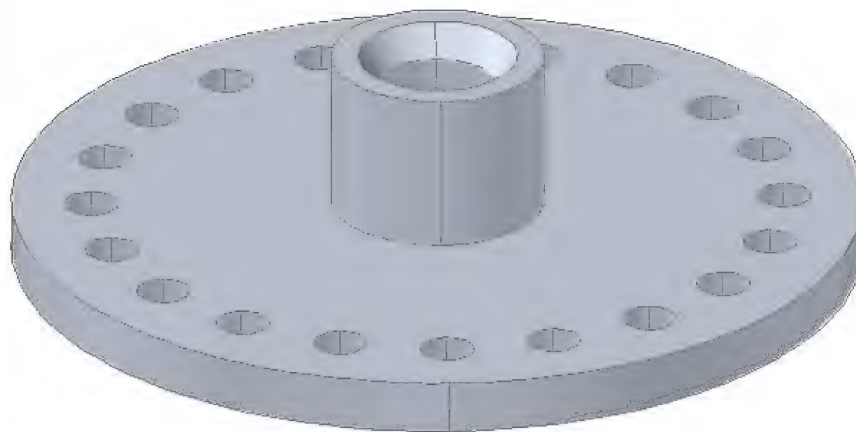


Figure 3.13.





## TUTORIAL 4: Creating Solid Model

### OBJECTIVES:

After completing this tutorial you should be able to:

1. Define, reorient, and utilize the User Coordinate System;
  2. Create a simple solid model from primitives and combine them using Boolean Operations;
  3. Use the ROTATE and MOVE commands to reposition parts for a model;
  4. Insert a model into the titleblock, scale it, and print it.
- 

### STEP 1

#### Creating 3D Drawings

Select **3D Modeling** from the Workspaces that you can see when you start AutoCAD 2013.

- a) Open a new drawing template from **File> New > Drawing**.
- b) Locate your **Titleblock.dwt template** from the “Select Template” dialog box.
- c) **Save As** the file as **Tutorial 4** inside your own folder.
- d) Make sure you are in **3D Modelling** Workspace and follow these steps:
  - i. Command : **VPOINT**  
Current view direction: VIEWDIR=0.0000,0.0000,1.0000
  - ii. Specify a view point or [Rotate] <display compass and tripod>: <Type **r** to select the **rotate** option and press **Enter** >.
  - iii. Enter angle in XY plane from X axis <270>: **300** <Type the desired angle in the XY plane>
  - iv. Enter angle from XY plane <90>: **35** <Type desired angle from the XY plane>



Figure 4.1

**Note:** Notice that the UCS Icon changed.



**STEP 2****Creating a 3D Box**

Command: **BOX** or Click Box from Home panel. Refer Figure 4.2



**Figure 4.2**

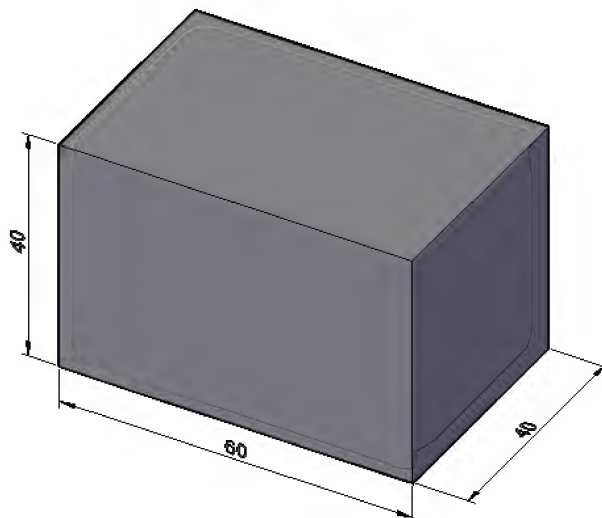
Specify corner of box or [Center] <0,0,0>:

- Use a **coordinate** or an **Osnap** to locate the first corner of the box on the XY plane.

**Specify corner or [Cube /Length]: type 60,40 and enter.**

**Specify height:** Type a height value (positive or negative). Type **40**

- The Box should appear as in Figure 4.3.



**Figure 4.3**

Use the **box** command again, but instead of drawing the box from the corner to corner, you will **add it about a center point**.

To do so, you must **add 3 construction lines inside** the first box same as Figure 4.4.

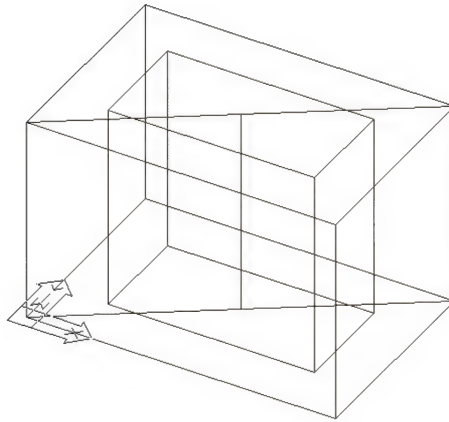




Figure 4.4

- Command: **Box** 
- Specify corner of box or [Center]<0,0,0>: type **ce**
- Specify center of box<0,0,0>: Use the **midpoint symbol**  to locate the center of this box at the midpoint of the vertical construction line you just added.
- Specify corner or [Cube/Length]: type **@-20,-10,20**.
- **Erase** the construction lines you used to locate the center for the second box.


To complete this drawing, **subtract** the **small box** from the **large box** using the **Subtract** command. Refer **Step 4 on Tutorial 3** to do subtract and to change your Visual Style.



Figure 4.5

**STEP 3**

To add the next feature,

- You must **change the UCS (User Coordinate System)** so the **XY Plane** is oriented to the **Front face**.
- The first way is find **UCS, Named UCS** button;  under Coordinates panel from View tab.
- The second way is to type **dducs** at a **Command:** prompt, which will bring up the UCS dialogue box. See Figure 4.6. Now, select the **Front UCS** orientation in the UCS dialogue box. Then **Set Current** Button. Then click **OK**. Look at the **UCSICON** on the AutoCAD screen to be sure the UCS changed.

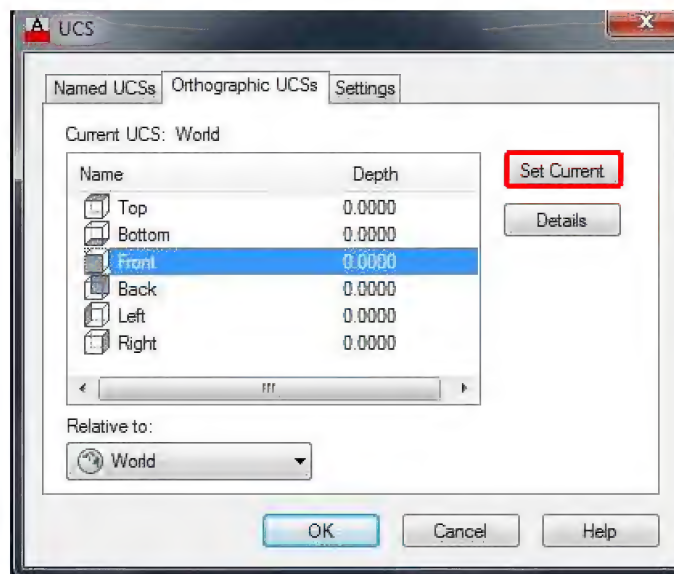



Figure 4.6

**STEP 4**

With the **UCS** oriented to **Front**, you can add a cylinder, with the **CYLINDER** command , through this side of the model.

**Note:** The UCS change was needed before you could add this cylinder because the **circular base of a cylinder is always oriented with the XY plane of the current UCS**.

To locate the cylinder,

- Add a **construction line diagonally across the front surface of the box** (as seen in Figure 4.8). The cylinder will be located at the **midpoint** of this construction line.

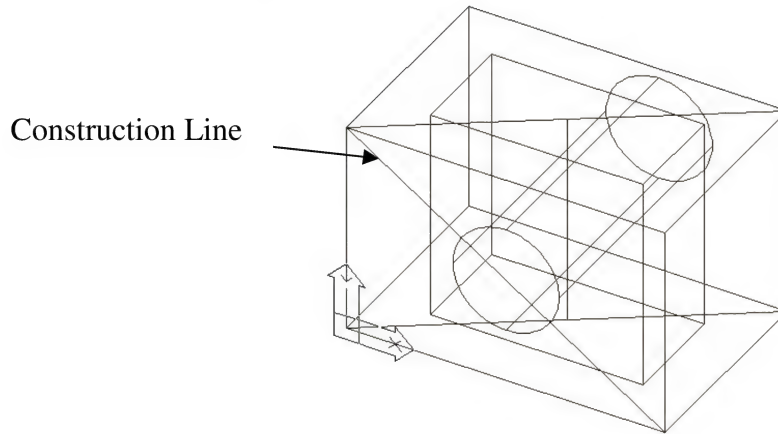


Figure 4.7

- Now, **add a cylinder** at the **Midpoint of the diagonal construction line** that has a **radius of 10** and a **height of -50**.

\* **Notice the use of the negative number for height.** The cylinder height, which is **along the Z-axis**, must be negative so it will be projected into the model. A positive height value would project the cylinder forward so that it sat on the front of the box's face.

- After adding the cylinder, **erase the diagonal construction line**.
- Use the **Subtract** command to remove the cylinder from the rest of the model. See Figure 4.8.

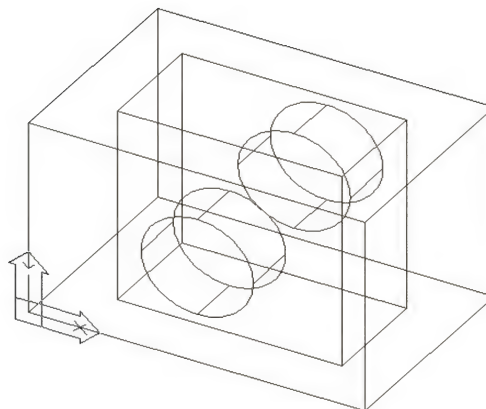
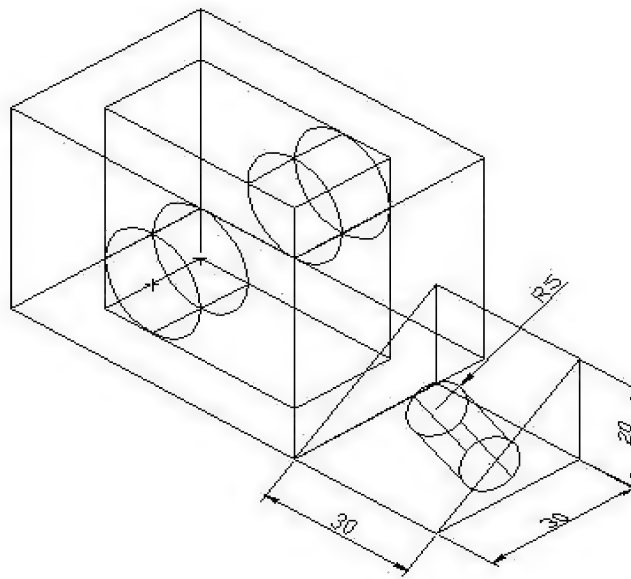


Figure 4.8


**STEP 5**

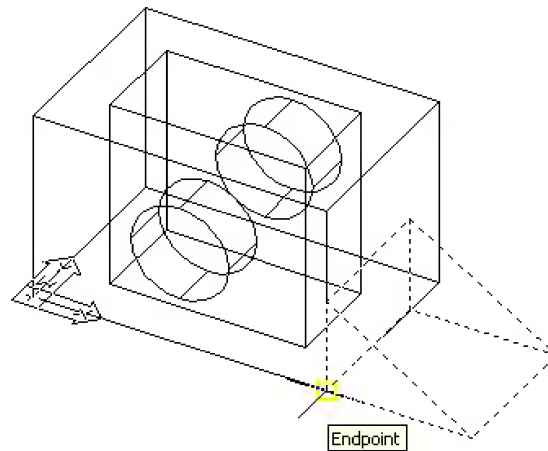
To carry out the next step, use the **UCS** command to change back to the **World Coordinate System**.

- You can make this change by typing **ucs** at a **Command:** prompt, and selecting the **World** option.
- The last primitive you will add to the model is a small **Wedge** on its **RIGHT FRONT CORNER** as in **Figure 4.9**.




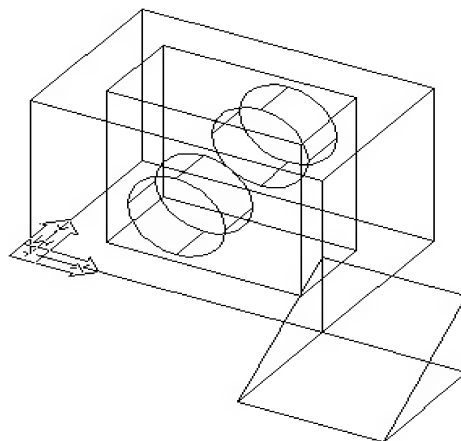
**Figure 4.9**

- The **Wedge** command will prompt you for the two diagonal corners of the base of the wedge.
- The next prompt will ask for the wedge's height. Wedges tend to be oriented in one direction. Therefore, you will create the wedge, rotate it into a new position, and then move it to its final location.
- With this information, activate the **WEDGE** command . Locate the first corner of the wedge at the Right Front corner of the existing model. When prompted for the next corner, use a **relative coordinate** to place this corner at **30,30** and **20** for the height. See Figure 4.10.


**Figure 4.10**

The **Wedge** you just added is not in the orientation that is needed, so you need to **ROTATE** it and then **MOVE** it into the correct position.

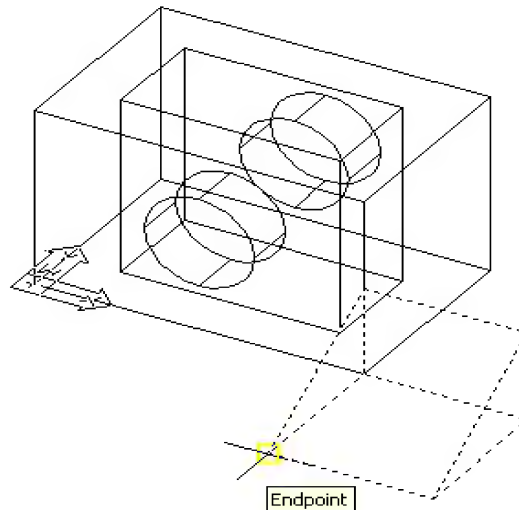
- Use the **Rotate;**  command to turn the wedge **-90 degrees**.
- A prompt will instruct you to select a **Base point**. The **Base point** is the position on the wedge that you will rotate the object around. You should use the **Endpoint Osnap** and the front left corner of the wedge as its Base point. See **Figure 4.10** and **Figure 4.11** as your references.
- Finally, the prompt will ask you for the degrees of rotation. Use **-90** so that the figure will rotate into the correct position. **Note:** You could also have used 270 degrees to rotate the object to the same position.
- Your figure should look like Figure 4.11 when you are finished.

**Figure 4.11**



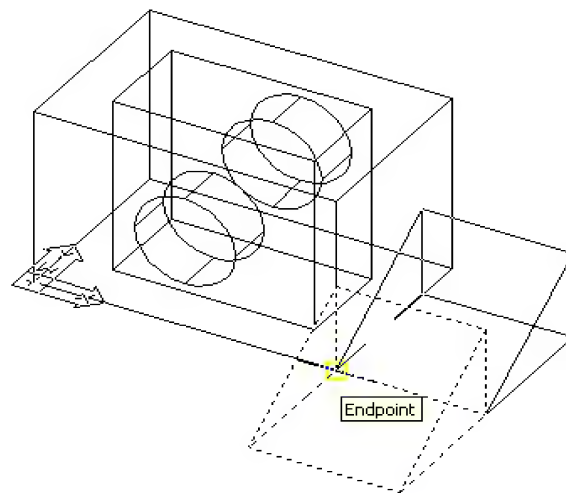
To change its position, you use the **MOVE** command 

- Select object you want to move (the wedge).
- Specify base point: select the **Front LEFT corner** of the **wedge**. See Figure 4.12.



**Figure 4.12**

Finally, select the **Front LEFT corner** of the **box** as the point to align the wedge with. See **Figure 4.13**.



**Figure 4.13**

The **final figure** should look like the one in **Figure 4.14**.

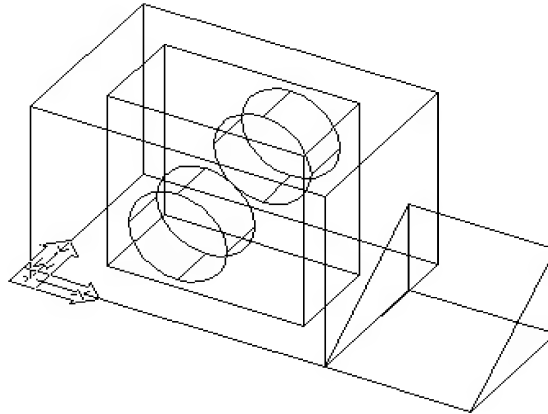


Figure 4.14

**STEP 6**

To finish the model, you will add a cylinder to the middle of the sloped surface of the wedge. Because the base of a cylinder is always aligned with the XY plane, and we want to have the cylinder parallel with the top of the wedge, you must **reorient the UCS to the face of the wedge**.

- Activate the **UCS** command; select the **New** option and then **3point**.
- **Look at** Figure 4.15, when the prompts ask for a new **origin point**, use an **Osnap** to select the point labeled as **1**. When asked for a positive on the new **X-axis**, select the point labeled **2**, and when ask for a positive position on the **Y-axis**, select the point labeled **3**. When you are finished, the **UCS** will be oriented to the figure as it is in Figure 4.15.

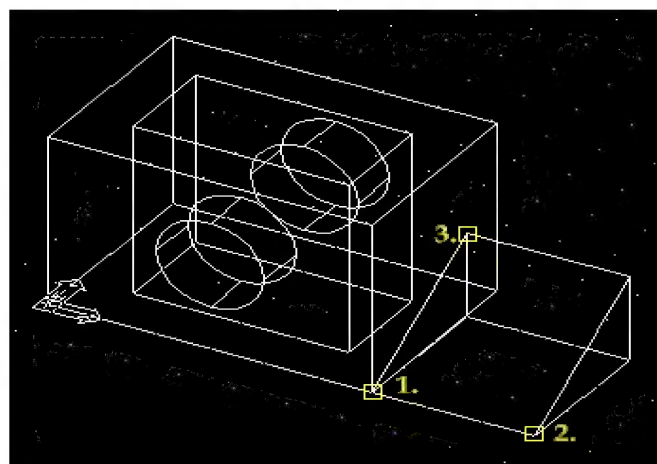
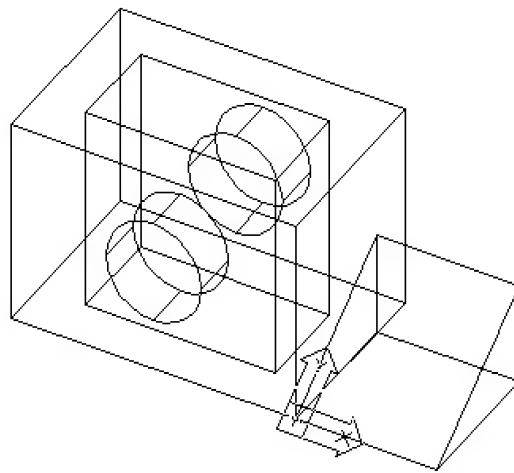
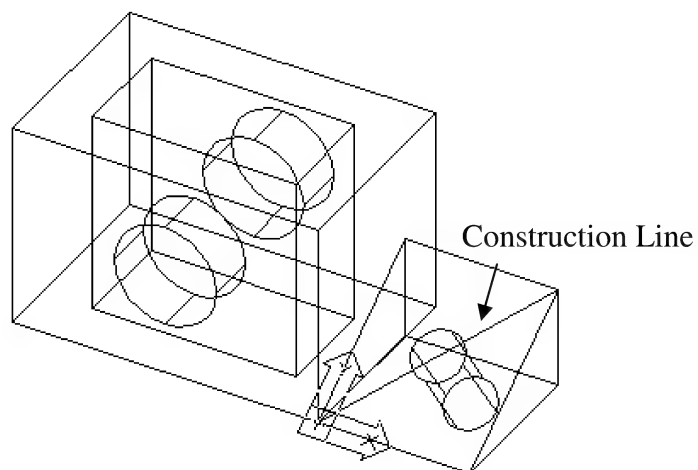


Figure 4.15


**Figure 4.16**

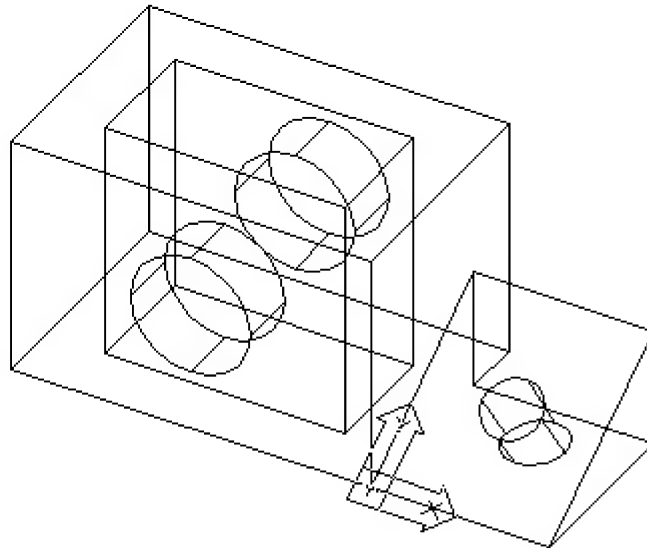
With the UCS aligned with the slope of the wedge, construct a **line** from **one corner of the sloped surface** of the wedge **diagonally** to the **opposite corner** of the same surface. See Figure 4.17. Use the **Midpoint** of this construction line to place a **cylinder** that has a **radius of 5** and a **height of -20**.

**Figure 4.17**

Once the cylinder is placed, **Subtract** the **cylinder from the wedge** . **Erase** the construction line used to place the cylinder.

To finish the figure (see Figure 4.18), you will use the **UNION** command to connect the wedge to the rest of the model.

- To access the **Union** command you can type **union** or **uni** at a prompt command, OR select the **Union** button  on **Solid Tab** under the **Boolean Panel**. **Select both object** and then press **enter**.



**Figure 4.18**

## **STEP 7**

With the model complete, you are ready to insert it into your titleblock.

- Move to **Layout1** by selecting the **Layout1** tab. If the model appears in the layout, click on the viewport edge and erase it.
- Next, use the **VPORTS** command to create a **Single Viewport** that fills the drawing area of the titleblock.
- In Layout1, click on the edge of the viewport to select it. **HINT:** Remember you can click on the upper edge of the titleblock to select this viewport because it overlaps the edges of the titleblock image area.
- Now, change a scale to **2:1** in the **Scale** window to scale the model size. Finally, if the model needs moving, use the **PAN** command to adjust its position in the window.

- You must **Explode** the titleblock before you can edit the text if it is not already exploded. Use **ddedit** to **edit the text** in the block as needed. Remember, you need to change the size in the **SCALE:** section to match the size of the figure inside the titleblock or **2:1**. See Figure 4.19.

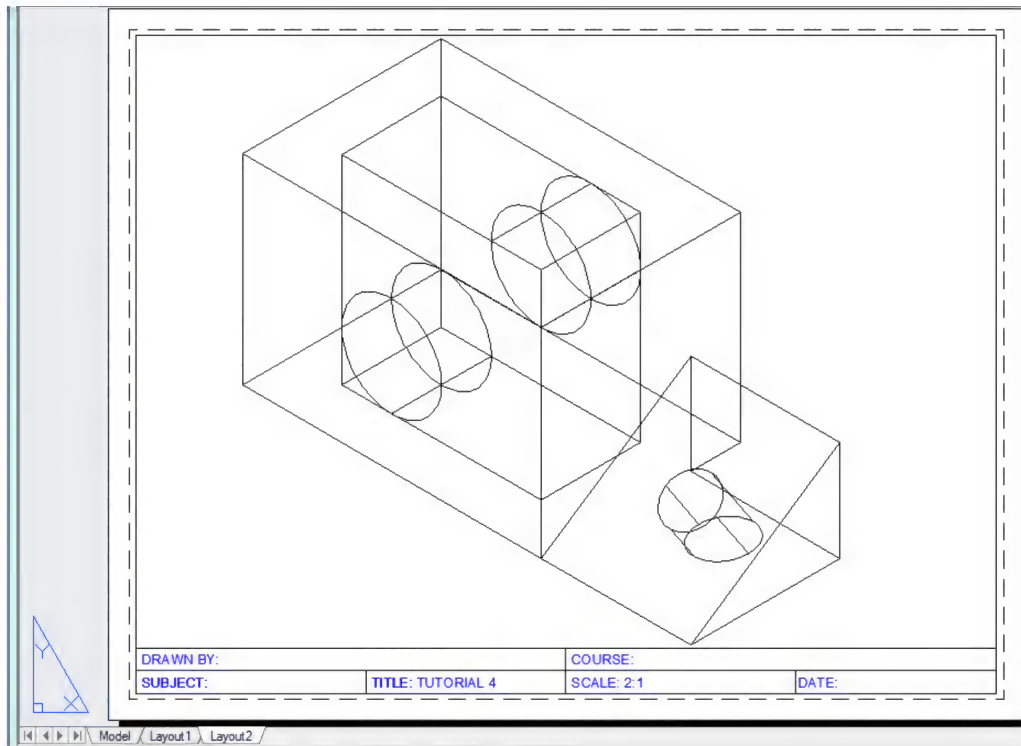


Figure 4.19



## TUTORIAL 5: Orthographic and Isometric

### OBJECTIVES:

After completing this tutorial you should be able to:

- 1) Use the User Coordinate System;
  - 2) Convert a solid model to a multiview using the SOLVIEW and SOLDRAW commands.
- 

### 5.1 Introduction

A view of an object is known technically as a projection. A projection is a view conceived to be drawn or projects onto a plane known as plane of projection. A system of views of an object formed by projectors from the object perpendicular to the desired plane of projection is known as **orthographic** or **multiview projection** which is the standard method of representing objects for manufacturing processes.

An orthographic one of method used to show the real shape of an object at one plane. Orthographic projection involves viewing an object from different directions – from the **front**, **side**, **top** or from any other viewing position. Orthographic projection often involves:

- ✚ The drawing of details which are hidden, using hidden detail lines;
- ✚ Sectional views in which the article being drawn is imagined as being cut through and the cut surface drawn;
- ✚ Centre lines through arcs, circles spheres and cylindrical shapes.

### STEP 1

- 1) Open your Tutorial 4 and save as **Tutorial 5** into your own folder.
- 2) **Erase the viewport** containing the drawing (Make sure you are in paper side of the layout) and change to **Model** space (drawing area) using **Model tab**.
- 3) Change the ucs position to World Coordinate Systems (WCS). Type the **ucs** command, choose **W** and press Enter for World Coordinate Systems (WCS).
- 4) Now you will be introduced to **Tiled Viewports**. With the **vports** command, create **4 vports** by choosing **Four: Equal** from the Viewports dialogue box.

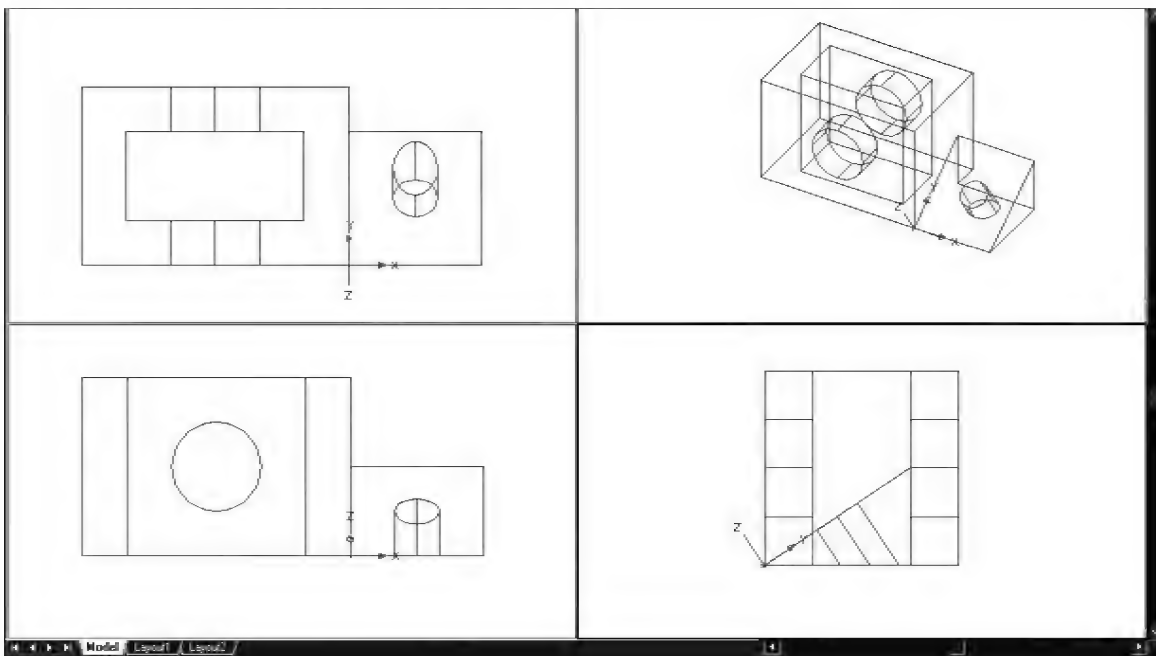
**Note :** **Tiled Viewports** allow you to see multiple views of the **SAME model**.



**STEP 2**

The next step is to change the **Vpoint** in three of these viewports, so that you have a **Top**, **Front**, and **Right side** view of the model.

- 1) To create a **TOP View** of the Model:
  - a) Click the **Top left** viewport.
  - b) Activate **vpoint** and select the **ROTATE** option.
  - c) Angle IN XY Plane: **270**
  - d) Angle FROM XY Plane: **90**
- 2) To create a **FRONT View** of the Model:
  - a) Click the **bottom left** viewport.
  - b) Activate **vpoint** and select the **ROTATE** option.
  - c) Angle IN XY Plane: **270**
  - d) Angle FROM XY Plane: **0**
- 3) To create a **RIGHT SIDE** view of the Model:
  - a) Click the **bottom right** viewport.
  - b) Activate **vpoint** and select the **ROTATE** option.
  - c) Angle IN XY Plane: **0**
  - d) Angle FROM XY Plane: **0**
- 4) Look at Figure 5.1 to check your orientations.



**Figure 5.1:** View orientations.

### **STEP 3**

#### **Solview Command**

The solview command will create layout of view based on a series of prompts, depending on the type of view (top, front and side) that user want to create. Usually, **the initial view that serves as the starting point for other orthogonal views is base on the current user coordinate system**. This need to be determined before beginning of this command. Once an initial view is created, it is very easy to create ortho, section and auxiliary views.

#### **Initialize A UCS With View Option**

- 1) Click the **bottom left viewport** (Front view) to set it as the initial view.
- 2) Command : **ucs** < press enter>
  - a) Enter and option [New/Move/ orthoGraphic /Prev/ Restore/Save/Del/ Apply/?/World] <World>: **n** <type n for the New option and press enter>
  - b) Specify origin of new UCS or [Zaxis/ 3point/ OBject/ Face/ View/X/Y/Z <0,0,0>: **v** <type v to select the View option and press enter>

The **UCS** will change to **match the current Viewpoint (Vpoint) configuration** of the whole screen or the active viewport.

- 1) **Activate the SOLVIEW command**, it will automatically switch to PAPER SPACE (Layout1) if you are not currently in it.
- 2) Select the **UCS** option.
- 3) When the prompt asks for a **UCS**, choose **current** and press **Enter** to use the **Front UCS** you have already set.
- 4) When the prompt asks for a **scale factor**, type **1** (same size).
- 5) When the prompt indicates “**Specify view center**”, click on the screen **in an appropriate position for a front view**. A front view of the model will appear on the screen. If the front view is not in a good location, you can click more than once until you think you have it in a good location.
- 6) When you have the view positioned, press **Enter** until the prompt directs you to “**Specify first corner of viewport**”.
- 7) When asked for the first corner, **click and release the LEFT mouse button in a position that would indicate one corner of a rectangular opening** that would contain the model view. See **Figure 5.2**.

- 8) When the prompt directs you to "**Specify opposite corner of viewport**", complete the rectangular viewport by clicking your mouse to the opposite diagonal corner of the rectangle.
- 9) Enter view name as **FRONT**.

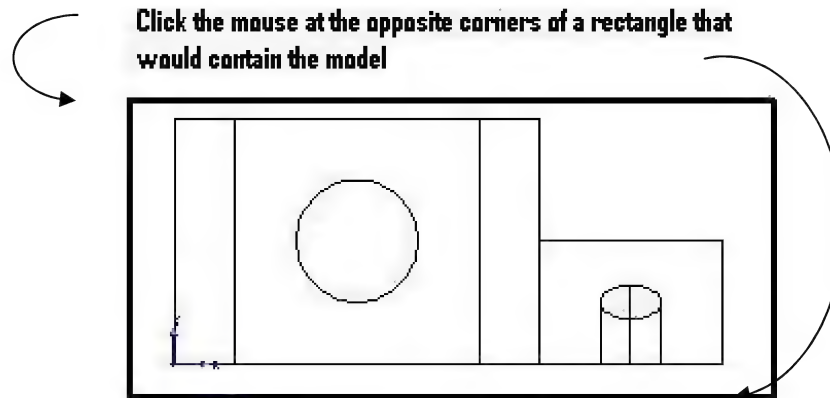


Figure 5.2: Front View

#### To place the TOP VIEW:

- 1) **Stay in the Solview command**, or return to it.
- 2) This time select the **Ortho** option.
- 3) Select the **TOP EDGE** of the **FRONT** viewport with the **Left mouse button** to indicate the "**Specify side of viewport to project.**" See Figure 5.3.
- 4) Place the view in an appropriate position for a **Top** multiview.
- 5) Name the view as **TOP**.

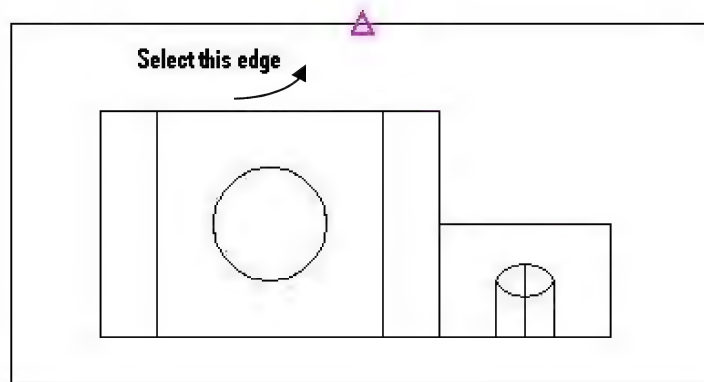


Figure 5.3

**You must click on the APPROPRIATE edge of the viewport**, when using the **Ortho** option. For example, if you click on the bottom edge of the FRONT viewport, AutoCAD will allow you to place the view in the TOP position, but it will be a **Bottom view** rather than a Top view.

Now, repeat the steps using the **Ortho** option of **Solview** to create :

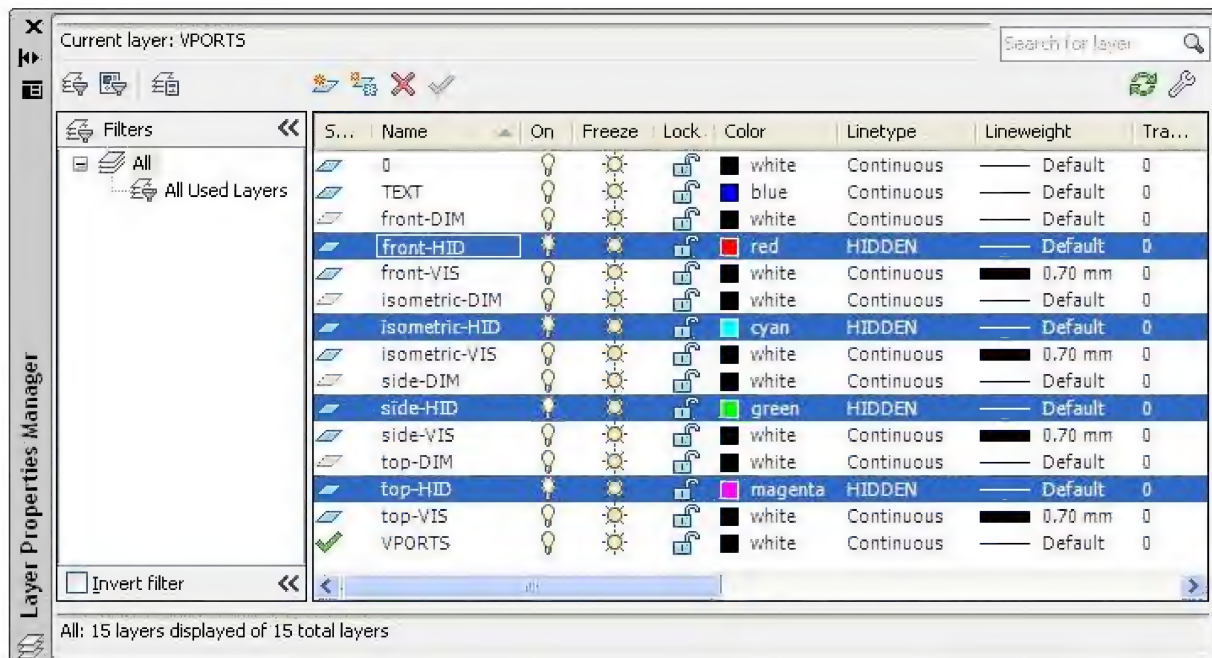
- 1) the **Right Side view** and name it as **SIDE**.
- 2) the **Isometric view** and name it as **ISOMETRIC**.
- 3) **Save** your file.

**Note:** Using Solview, you can display the SAME MODEL seen from FOUR DIFFERENT ANGLES in each **NAMED Viewport**.

#### STEP 4

The SOLVIEW command also automates a series of **viewport layers** for the **Hidden** and **Dimension lines**.

- 1) Click on the **Layer Properties** button to bring up the **Layer Properties Manager** dialogue box in Figure 5.4 below. In this dialogue box, you should find a series of **Layers** labeled **Front-DIM**, **Front-HID**, **Front-VIS**, **Side-DIM**, **Side-HID**, etc. These layers were added by the **Solview** command.
- 2) For now, we are only interested in the **Front-HID**, **Side-HID**, **Top-HID** and **Isometric-HID** layers. Assigned the **Hidden** linetype and have their **color** changed. Repeat this procedure for each of the layers ending in **-HID** (for **Hidden**).



**Figure 5.4:** Named Viewports layers created.

- 3) Change the **LINE THICKNESS** of the **-VIS** layers. Locate a **-VIS** layer and look at the setting under the **Lineweight**. It should be set to **Default**.
- 4) Click on the word **Default** and AutoCAD will open the **Lineweight** dialogue box. In this box, scroll down to **0.7 mm** and then click on the **OK** button. Repeat this for all of the **-VIS** layers.

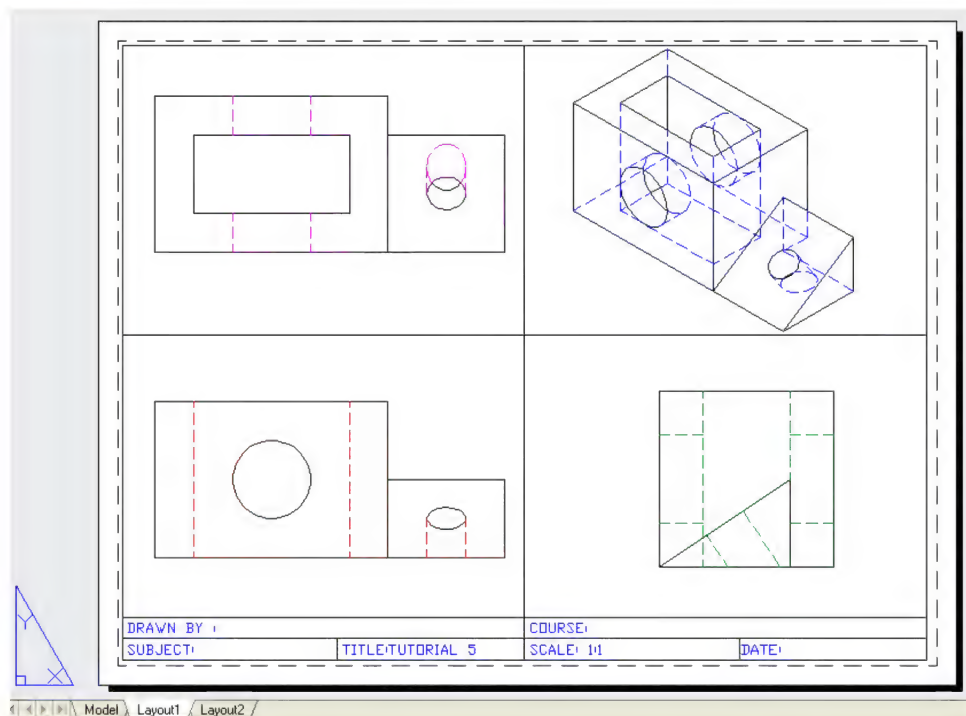
**Note :** Making this change will not show in the drawing until you print. If you do not make this change, the visible lines in the **top-VIS**, **side-VIS**, and **front-VIS** layers would be the same thickness as the center and hidden lines.

## **STEP 5**

### **Soldraw Command**

Soldraw command is used to draw or change the view once it has been laid out by solview command and **create 2D features** including hidden lines.

- 1) Close the **Layer** dialogue window.
- 2) Finally, type **SOLDRAW** at command line, and type **all** when asked to select object and press enter to finish the command.
- 3) **Save.** Your drawing should appear like Figure 5.5 when you finish it. The hidden lines in your drawing should now appear in the color you selected.



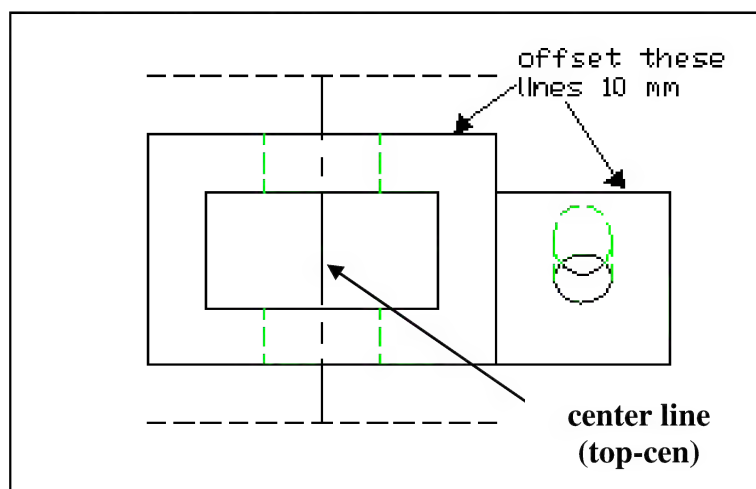
**Figure 5.5:** Hidden line appear after soldraw

**STEP 6**

The next step will be to add center lines to your drawing.

- 1) **Create three additional layers** named: **top-cen**, **front-cen**, and **side-cen**. Once they are added, **change the layer color assignment to something OTHER THAN BLACK** and the linetype to **center lines**.
- 2) Change to **Model side of the Paper Space**.
- 3) **Click in the Top viewport** to make it the **active window** and change to the **top-cen** layer.

**Note:** With the Top viewport active, you can now modify the 2D multiview projection to add center lines. The easiest method for adding the center lines is to use a combination of the **Offset** and **Extend** commands.



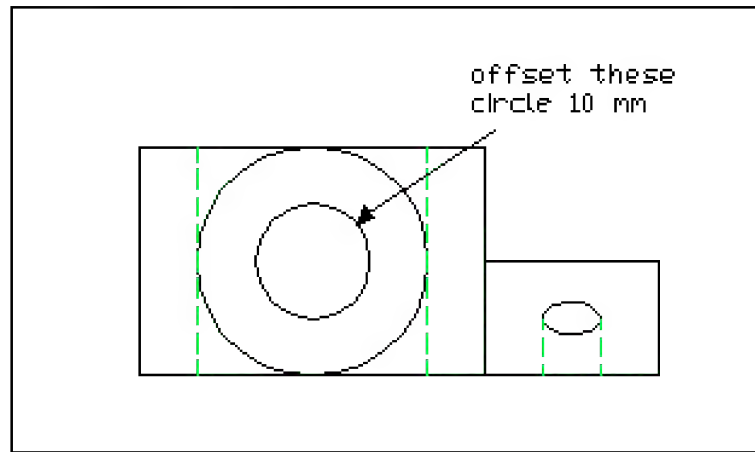
**Figure 5.6:** Adding center line

- 4) **FINISH ALL** the centre layer as in Figure 5.10.

**HINT:** The **standard rule** is that center lines should extend approximately **10 mm past the feature they are associated with** if the drawing is not dimensioned. Offset works with a circle or arc as easily as it works for a line. **Look at Figure 5.7 and 5.8.**

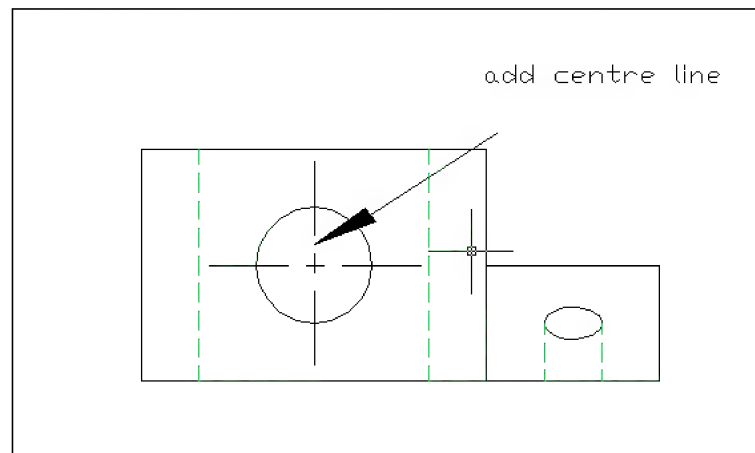
- 5) **Make sure** that you are in the **CORRECT LAYER** for the view in which you are adding center lines ( **Use front-cen** layer in the **Front viewport** and **side-cen** layer in the **Side viewport**).





**Figure 5.7:** Offset circle by 10 mm

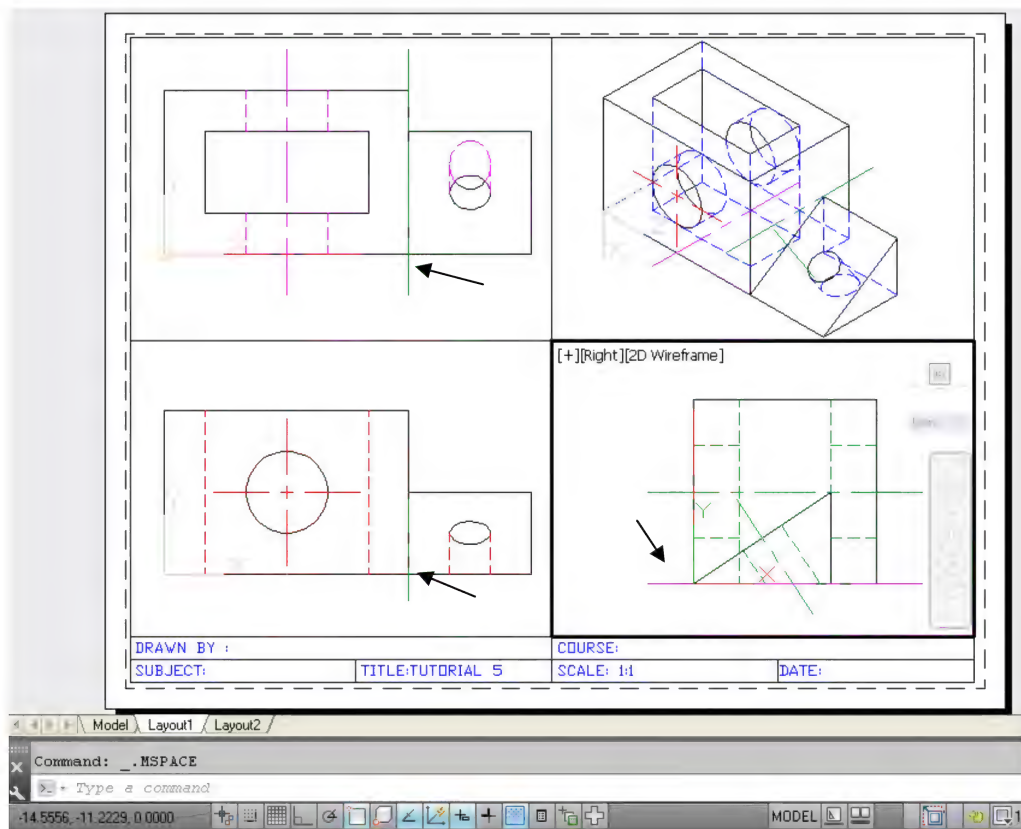
**Note:** If you try to place center lines so that they extend 10 mm past the circle, they will touch the visible lines around the circle. **THIS IS NOT GOOD DRAWING PRACTICE.** You **NEVER** draw center lines **so that they END at a visible line**. You either shorten or lengthen the lines when this situation occurs. To deal with this problem, **Offset** the circle **8mm** instead of **10**.



**Figure 5.8:** Adding center line

### **STEP 7**

Notice that the center lines you are placing in the Top view are also showing up in the other views. It illustrates that you are seeing both the model and the 2D projections of the model in all of the viewports. See Figure 5.9. We use **Vplayer (Viewport Layer)** to erase the extra center lines.



**Figure 5.9:** Center line from other view appear in viewport

The **VIEWPORT LAYER (Vplayer)** command provides control of the layers in a viewport when using multiple viewports in Paper Space. The **FREEZE** option under this command allows you to make specified layers in a specific viewport invisible.

Before activating the **Vplayer** command, you need to change to the **Model Side of Paper Space**. **Activate the Top viewport**. Use the **Freeze** option under this command to "**freeze**" the **front-cen** and **side-cen** layers. After you exit this command, the extra center lines in this view will disappear from this view.

- 1) Command: Type **vplayer** and **Enter**.
  - a) Enter an option: **f** < f for freeze option>
  - b) Enter layer name(s) to freeze : <Type **the name of a layer or layers** to freeze>
  - c) Specify viewport(s) <Current>: **<Enter>**
  - d) Enter an option : **< Enter to exit the command>**
- 2) Repeat the command for other layer until **center line from others view has been deleted**.

### 3) Wrong Lines Disappeared

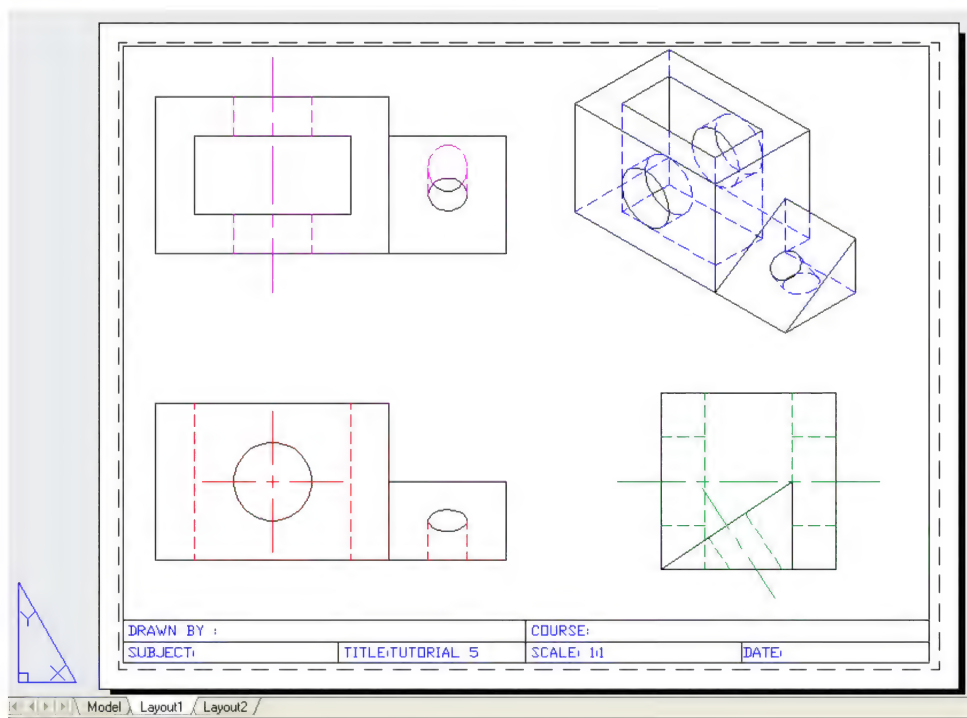
The **most common error** that causes this problem is that the center lines were added to the wrong layers. It is easy to forget to change the layer when you shift to a new viewport and begin adding center lines.

#### To Solve the Problems:

- First, **undo** until your entire center lines show again.
- Next, **select a center line** by clicking on it with the left mouse button. Look at the **Layer Status Window**. The layer that this center line was drawn on will appear in the window.
- If the center line is not on the correct layer, use the same **Layer Status Window drop-down menu** to select the proper layer while the line is selected. This will move the line to the correct layer.

**Note:** If the center line was already on the correct layer, hit **Escape twice** to completely deselect it before testing the next line.

**Switch back to PAPER side of Paper Space** and **use the Layer Status Window menu** to switch off **the light bulb icon to make the Vport layer invisible**. The borders around the views will disappear, and the drawing will look like **Figure 5.10** below.



**Figure 5.10:** Completed Orthographic Design



## TUTORIAL 6: Dimensioning

### OBJECTIVES:

After completing this tutorial:

1. You will be able to add horizontal, vertical, diameter, and radius dimensions to a multiview drawing in AutoCAD;
2. Create continue and baseline dimensions;
3. Create a dimensioning style; and
4. Use the Mtext option under the Dimension commands to create counterbore and countersink dimensions.

---

### STEP 1

Open your **Tutorial5** and **save as Tutorial6** into your folder (see Figure 6.1).

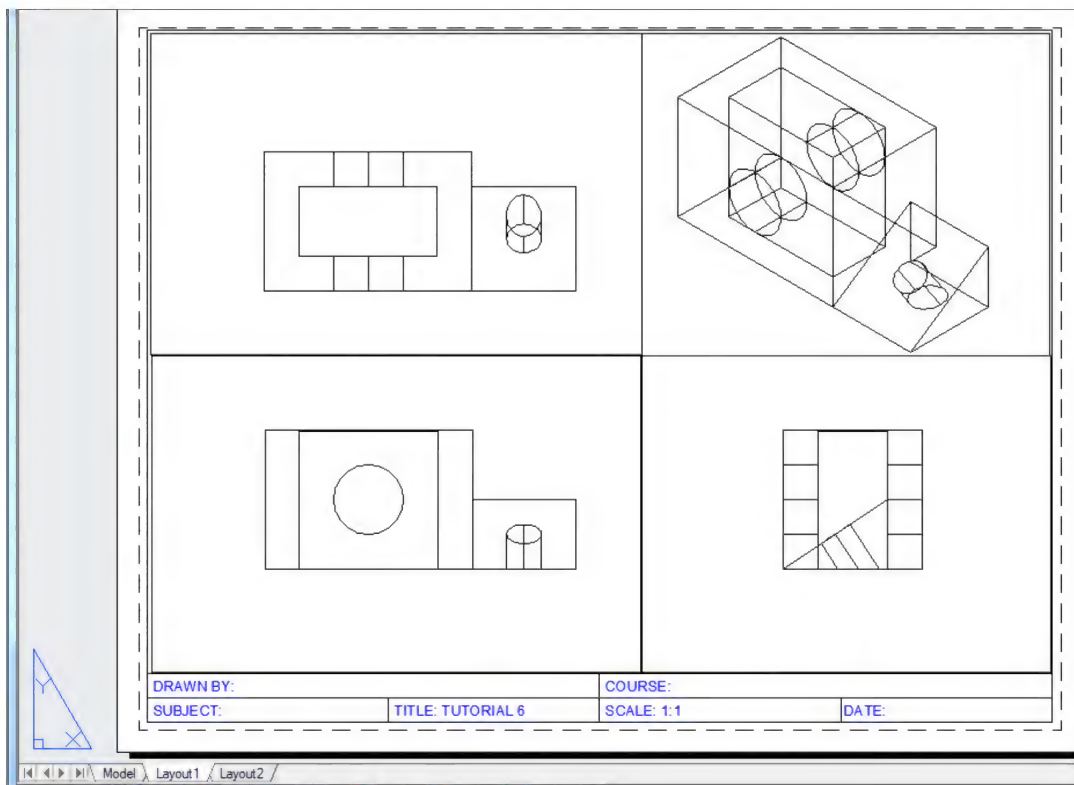


Figure 6.1

## STEP 2

### Dimension Panel

Dimension can be found on **Drafting & Annotation Workspace, Annotation panel** and click types of dimension that you want to use as Figure 6.2 or can be access from **3D Modelling Workspace >Annotate> Dimensions**.

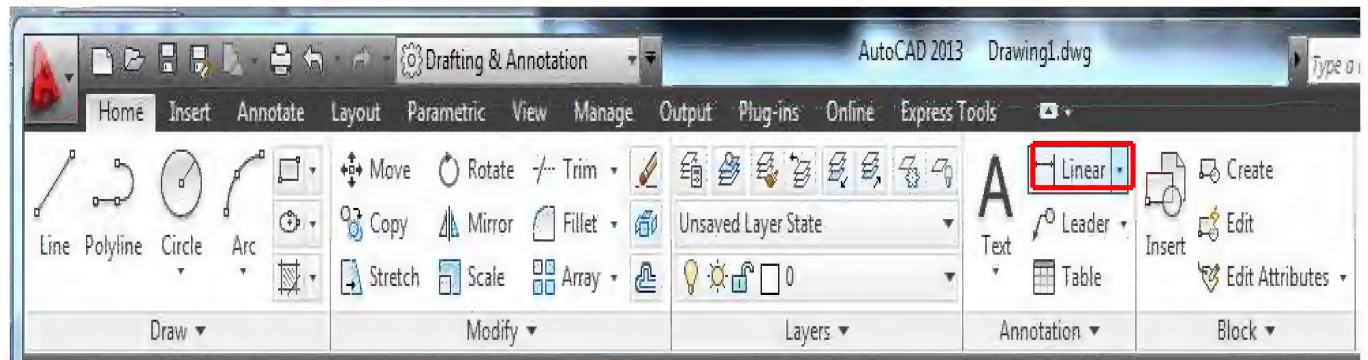


Figure 6.2

### Creating a Dimensioning Style.


Dimension styles can simplify dimensioning by predefining certain dimension formats. Click Dimension Style button  to have a **Dimension Style Manager**. If you using **3D Modelling Workspace** you must click the arrow near Dimensions to have a **Dimension Style Manager**. Refer **Figure 6.3** shows an example.

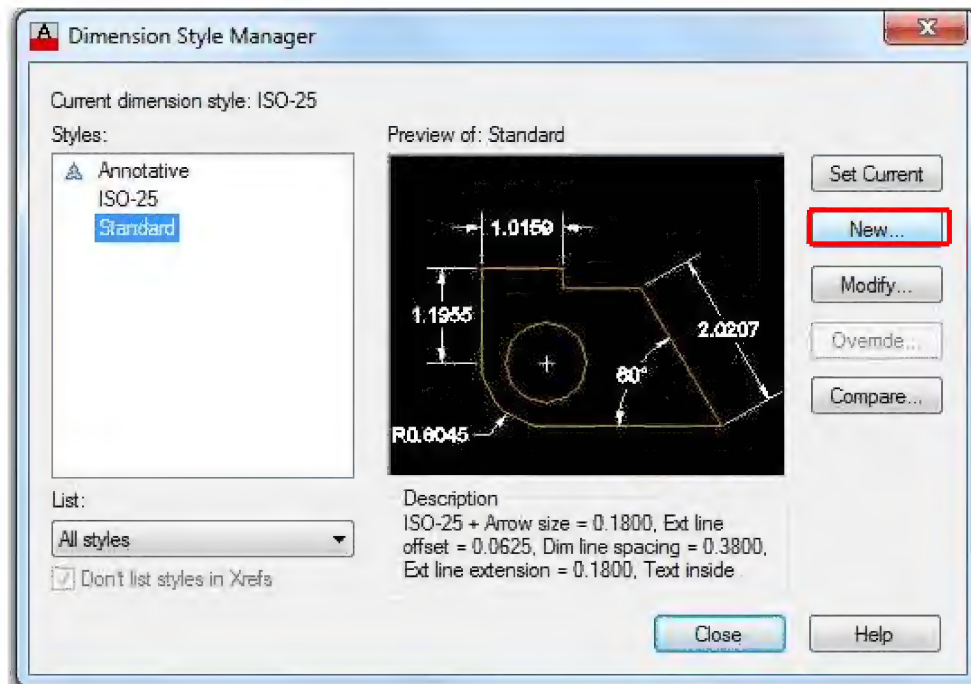


Figure 6.3

**On Dimension Style Manager** you can modify the dimension parameters, and create different dimensioning styles under a name of your choosing. These dimension styles can be defined and utilized later so that you do not have to set each dimension individually.

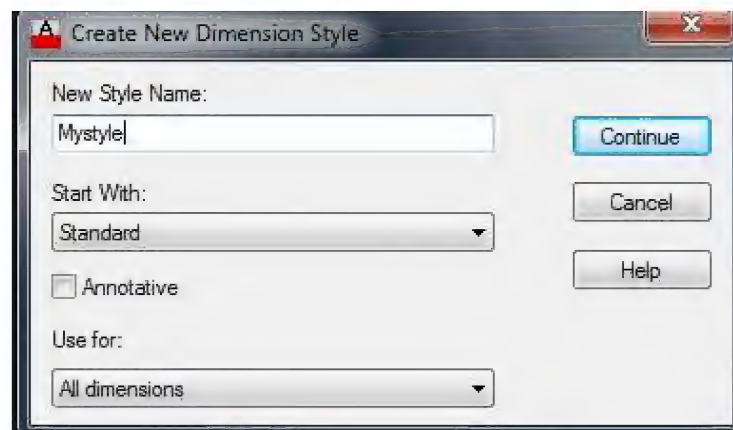
For this tutorial, you will create a style based on the **Standard** style, and save it under a different name.





**Figure 6.4 Dimension Style Manager**

To create your new dimensioning style, click on the **New** button. The **Create New Dimension Style** dialogue box will appear (Figure 6.5). In this box, you can begin the definition for a new style and if can select an existing style to base this new style on.



**Figure 6.5**

To create your new dimension style, type **Mystyle** in the window by the heading **New Style Name:**, make sure that Standard is selected in the window labeled **Start With:**, and then click on **OK**. The **Modify Dimension Style** dialogue box will appear.

The **Modify Dimension Style** has been designed so that you can define a variety of dimension parameters. When it first appears, the **Lines** tab should be showing (see Figure 6.6).

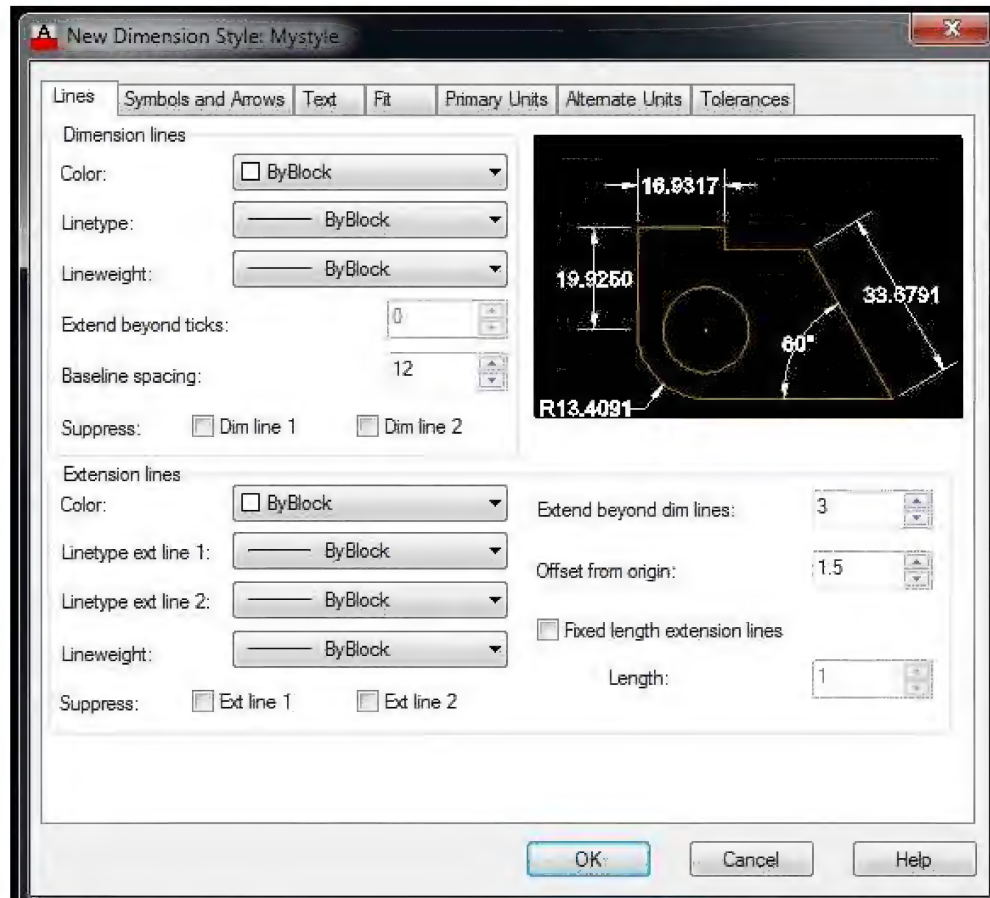


Figure 6.6 Lines Tab Setting

## LINES TAB

Under the **Dimension Lines** section— set **Spacing**: to **12** (this sets the space between stacked dimensions). **NOTE:** The minimum space between dimensions is 6mm; however, you will have some long dimensions and will be orienting your dimension text horizontally so you will need to have more space between the stacked dimensions.

Under the **Extension Line** section— set **Extend beyond dim lines**: to **3** (this indicates the distance the extension line should extend past the dimension line) and set **Origin Offset**: to **1.5** (this determines the gap between the extension line and the element being dimensioned).



## SYMBOLS AND ARROWS TAB

Under **Arrowheads** — set **Size:** to **3** (this sets the arrowhead lengths), and make sure that in the windows by the **1st**, **2nd**, and **Leader** headings they show **Closed filled**.

Under **Center Marks for Circles**— set **Size:** to **1.5** (this sets the length for half of the lines for a center mark).

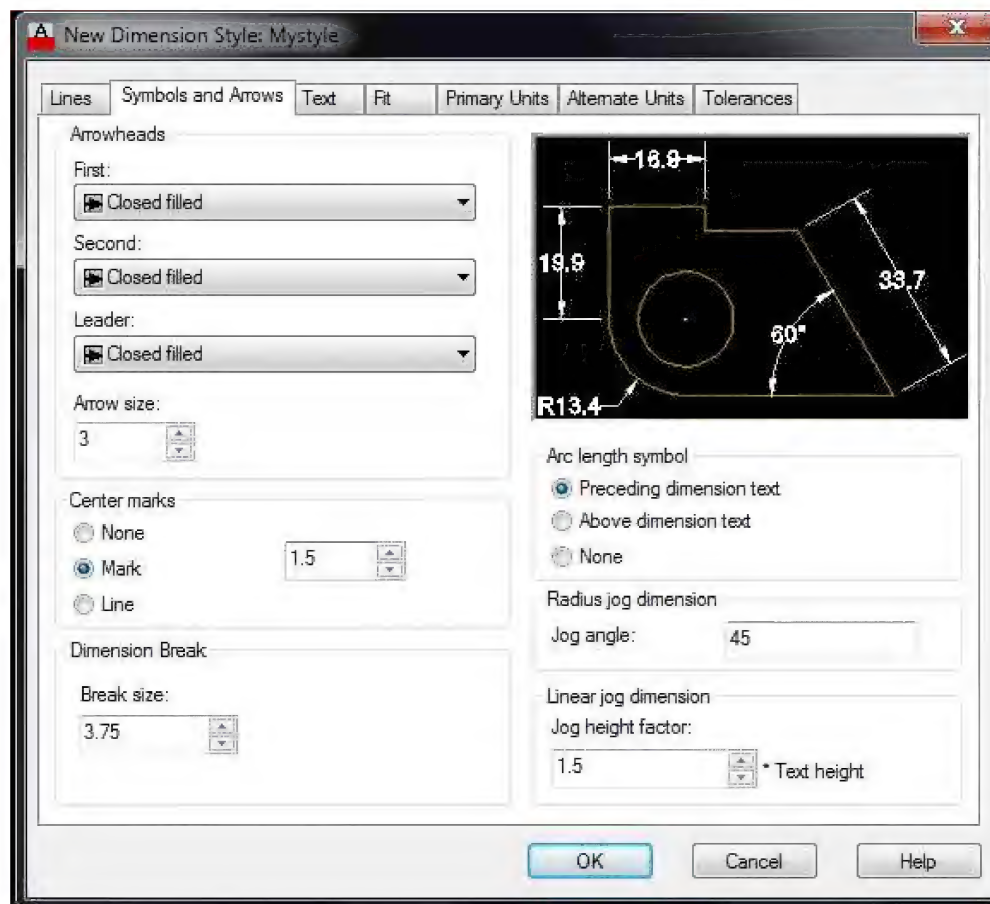


Figure 6.7 Symbols and Arrows Tab Setting

## TEXT TAB

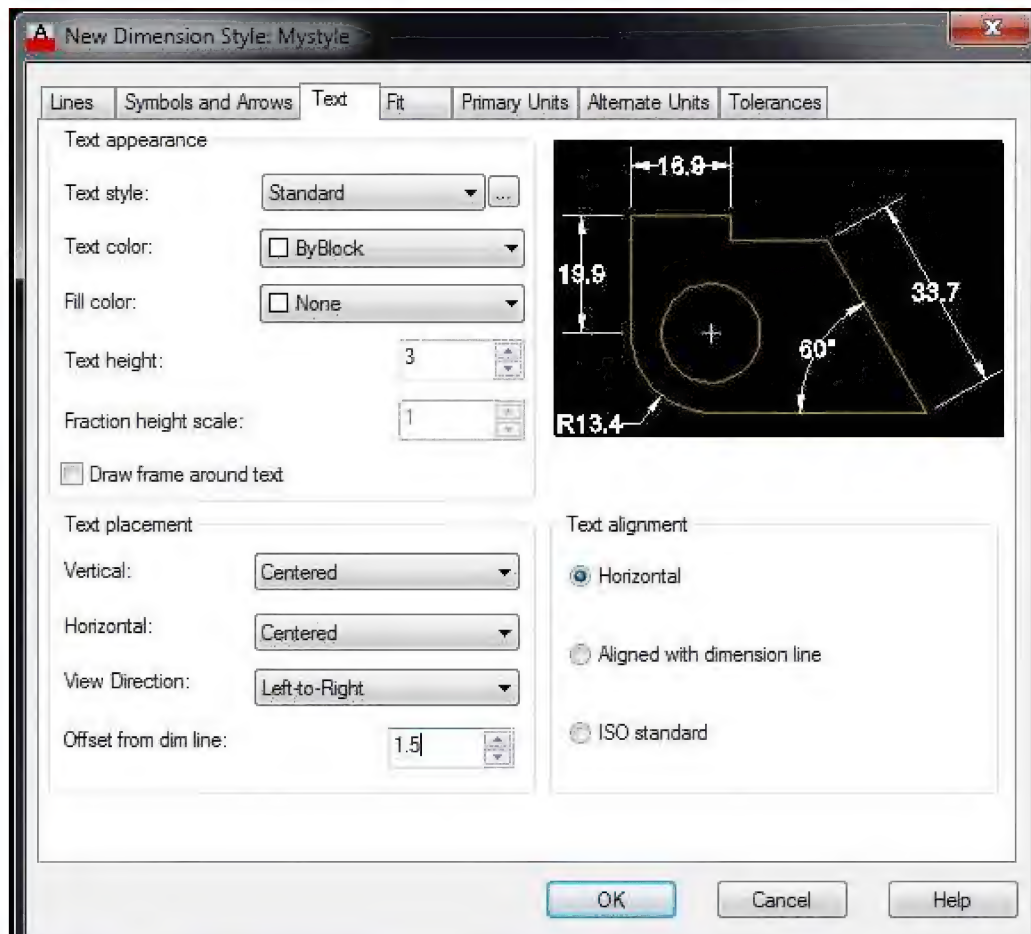
Under **Text Appearance** —Make sure that the **Text style:** is **Standard** and the **Text color:** is **ByBlock** (this allows the color to be controlled by the layers and assures that the font is the default).

Under the **Text Placement** section—Select **Centered** in the window by **Vertical:** and **Centered** in the window by **Horizontal:** (this orients the text to the dimension lines).

**Note:** If a dimension placed in this location does not work well with other dimensions, you can easily move a dimension to a new location after it is placed.

Under the **Text Alignment** section—Click on the **Horizontal** button (this orients all of the dimension numbers to a horizontal position).

In the window beside the heading **Offset from dim line:** type in **1.5** (this changes the size of the gap in the dimension line by establishing how far the dimension line should be from the text).



**Figure 6.8** Text Tab Setting

## FIT TAB

Under **Fit Options**—Click on the button next to the heading **Either the text or the arrow, whichever fits best** (this gives you the flexibility to have a dimension inside or outside of the extension lines).

Under **Scales for Dimension Features**—select **Scale dimensions to layout (paperspace)**. This scales the elements of the dimensioning system (arrows, numbers, etc.) to the scale used in paperspace.

Under **Fine Tuning**:—select **Place text manually when dimensioning** (this gives you the flexibility to place a dimension in the most appropriate position relative to other dimensions).

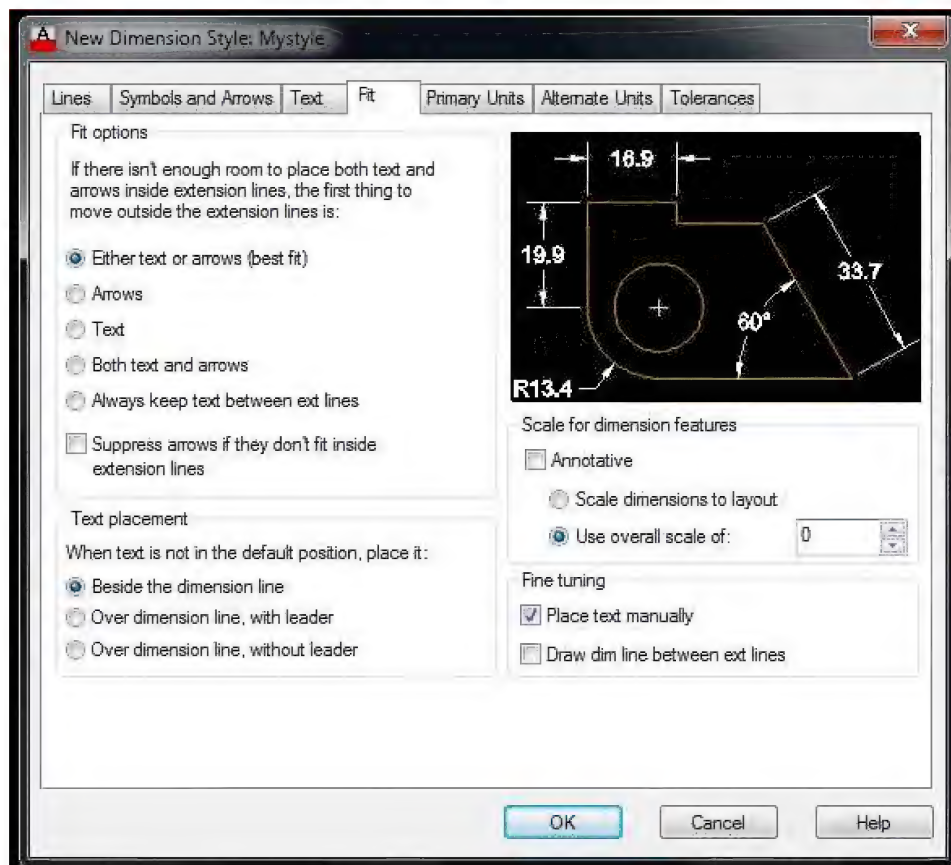


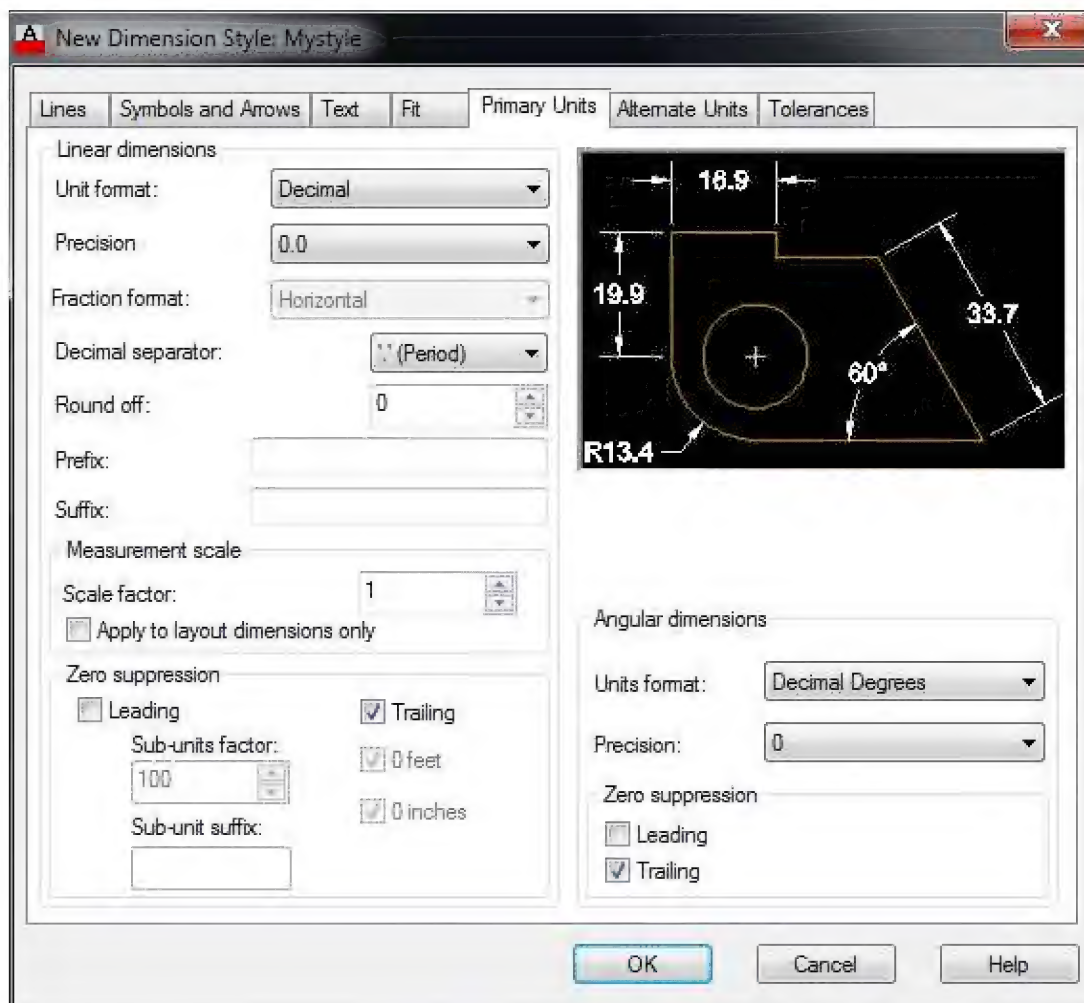
Figure 6.9 Fit Tab Setting

## PRIMARY UNITS TAB

In this box, make sure that **Linear Dimensions** are set to **Decimal**, **Precision** is set to **0.0** for one decimal point, and Round off is set to **0**.

Under **Measurement Scales**—leave the Scale factor at **1**.

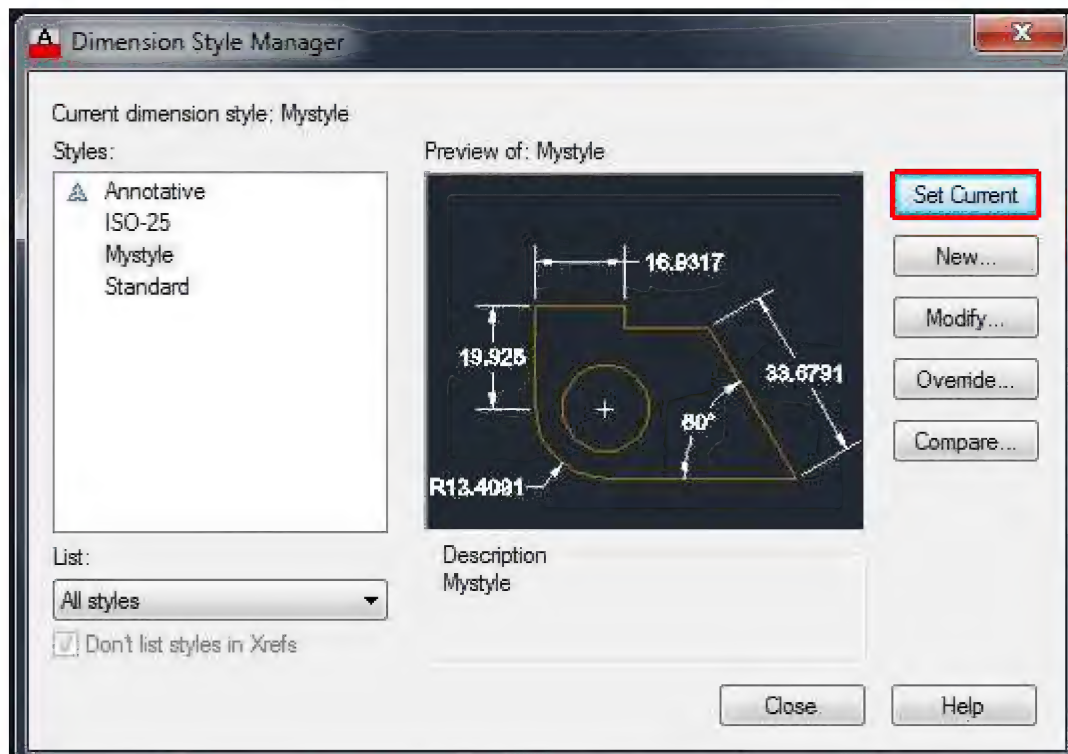
Beside **both** areas labeled **Zero Suppression**—click on the check box next to **Trailing** (this will remove any excess 0's after a number).



**Figure 6.10 Primary Units Tab Setting**

Click on **OK** to return to the **Dimension Style Manager**.

When the **Dimension Style Manager** box appears, make sure that **Mystyle** is still highlighted in the **Styles:** window and then click on the **Set Current** button to set this as your dimension style and then on close the window. See Figure 6.11.

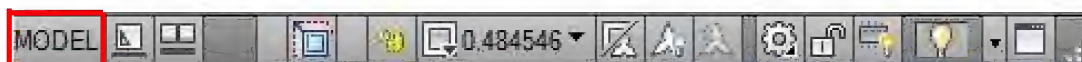


**Figure 6.11**

### STEP 3

## Adding Dimensions

Before adding your first dimension, make sure you are **in the MODEL side of Paper Space** and leave the **Vpport** layer **VISIBLE**. The **MODEL** button available at the bottom-right on the **Status Bar** same as **Figure 6.12**.



### Figure 6.12 Status Bar



Figure 6.13 shows you illustrations you will do. **Before adding this dimension**, make sure the **TOP View** active, change to the **Top-DIM** layer.

**Notes:** AutoCAD will not allow you to place dimensions in a view unless you have matched the layer to the view. If you attempt to add dimensions in a layer that does not match the view, the dimension does not work .

If you want a better view of the **TOP** view, change to the **PAPER** side of Paper Space by clicking on the **MODEL** button at the end of the Status Bar so that it changes to **PAPER** and use a **Window Zoom** to enlarge it on the screen. Change back to the **MODEL** side of Paper Space before adding the dimension and change the **layer** to **match the view**.

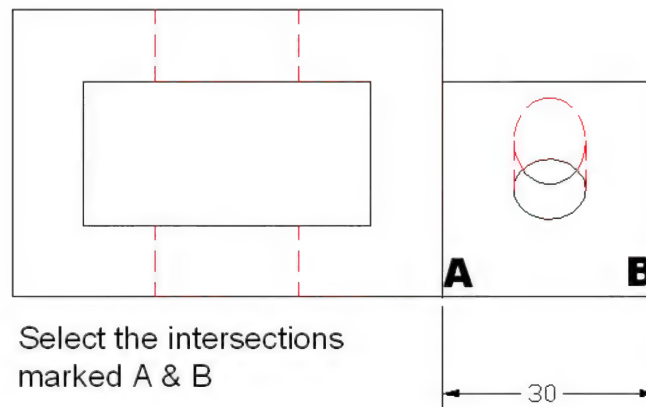



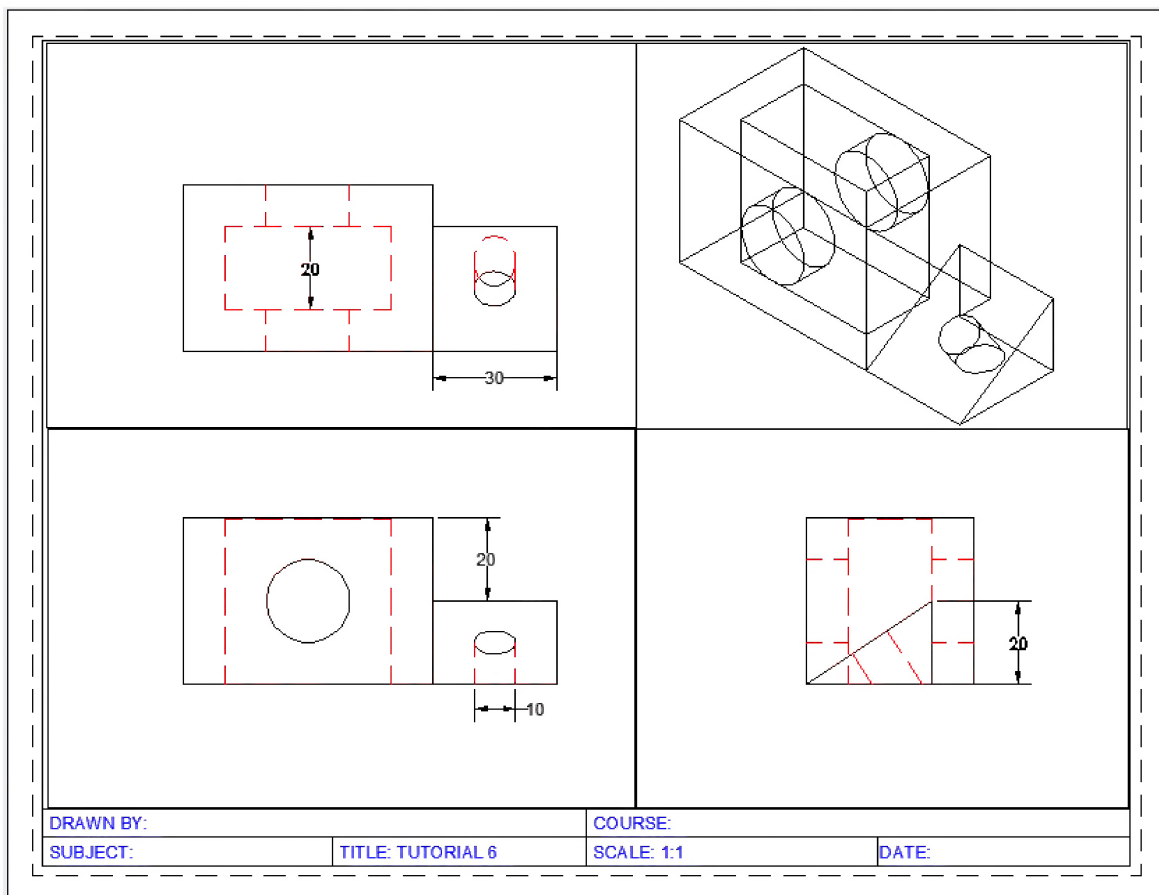
Figure 6.13

The dimension, shown in Figure 6.13, will be added with a [Linear](#) dimension.

- 1) Command : **<click the Linear dimension button;**  **>**
- 2) Specify first extension line origin or <select object>: **< Use the INTERSECTION Osnap and select position A (the first extension line origin) shown in Figure 6.13 >**
- 3) Specify second extension line origin: **< Use the INTERSECTION Osnap and select position B (the second extension line origin) shown in Figure 6.13>**
- 4) Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:  
Dimension text = 30 **< Place the location for the dimension text by clicking the Left mouse button on the screen >**
- 5) Now, use the same technique to place the rest of the dimensions seen in **Figure 6.14**

**Important Note:** Use the **Intersection Osnap** and not **Endpoint** to select the extension line origin points for dimensions in a 2D object. If you already have dimensions in place, you can **accidentally select the end of an extension line** instead of the object line.

**Note:** - Use **Osnaps** when placing dimensions to be sure they are accurate.



**Figure 6.14**

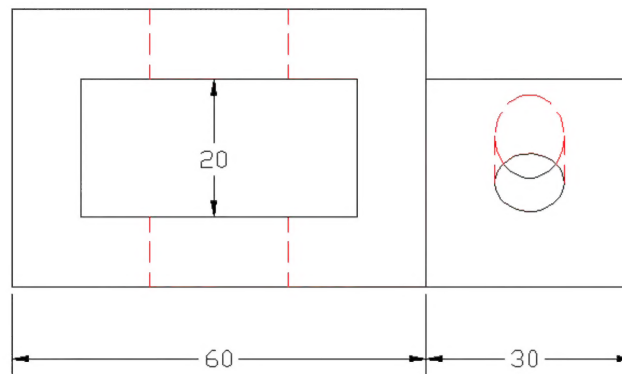
**ERASING A DIMENSION:** If you need to erase a dimension, the **Erase** command functions the same for dimensions as it does for other elements. Since dimensions are grouped elements, you can click on any part of the dimension, and the whole dimension selects.




**STEP 4****Continue Dimension**

A **Continue** dimension allows you to place dimensions in a continuous row so they are aligned to each other and share an extension line.

Look at Figure 6.15 to see where the first **Continue** dimension should go. You may want to change to **PAPER** side of **Paper Space** and **zoom in**, but remember to change back to the **MODEL** side of **Paper Space** before dimensioning.



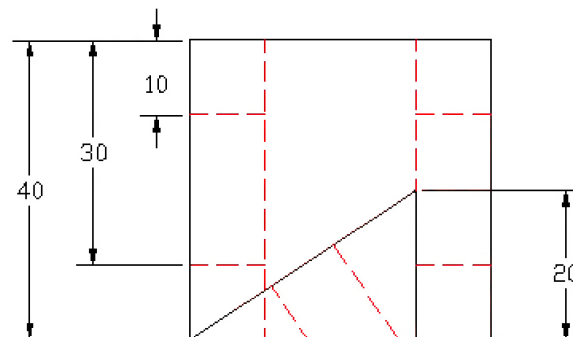
**Figure 6.15**

- 1) Command: **< select the Continue dimension button;**  **>**
- 2) Select continued dimension: **<Click on the bottom horizontal dimension (size is 30), in the top view, that you already added to the drawing (Figure 6.15)>**  
**Note:** If you were continuing the dimension you had just added, this prompt will not appear and only the next prompt is shown.
- 3) Specify a second extension line origin or [Undo/Select] <Select>:  
 Dimension text = 60 **< Click the intersection Osnap and select the right bottom corner of the rectangular slot (for the 60 dimension)>**
- 4) Specify a second extension line origin or [Undo/Select] <Select>: **<Press Enter until you end command>**

**Note:** If you get are trying to add a continue dimension to a dimension other than the last you added, you would use the **Select option** by typing **s** and then pressing Enter. AutoCAD would then give you the prompt: **Select continued dimension:** . Repeat the same steps to add the continued dimensions.

**STEP 5****Baseline Dimension**

Look at Figure 6.16. This figure shows the RIGHT SIDE view with the baseline dimensions added. Change the layer to **side-DIM**. A **Baseline** dimension allows you to stack dimensions off of a common point of origin. The dimensions of **30** and **40** are **Baseline** dimensions in this illustration.

**Figure 6.16**

Begin by adding a **Linear** dimension (**10**) for the rectangle. Add a **Baseline** dimension.

- 1) Command: `_dimbaseline` < **Click on the Baseline button;**  >
- 2) Select base dimension: < **Click on the dimension text of the 10**>



**Note:** Like the Continue dimension, the baseline dimension will only ask you to specify an existing dimension as a base if you are not using the last dimension you placed as the base dimension. When you change viewports, AutoCAD will prompt you to select this dimension.

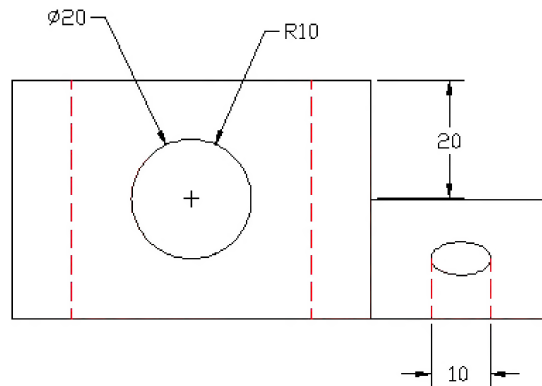
- 3) Specify a second extension line origin or [Undo/Select] <Select>: `_int` of  
Dimension text = 30 < **Select the endpoint of the vertical line on the right edge**  
**(shown for the 30mm dimension in Figure 6.16)** >.

**Note:** The dimension appears and is stacked outside of the 10 dimension.

- 4) Repeat for the second Baseline dimension of **40**.
- 5) Save your drawing.

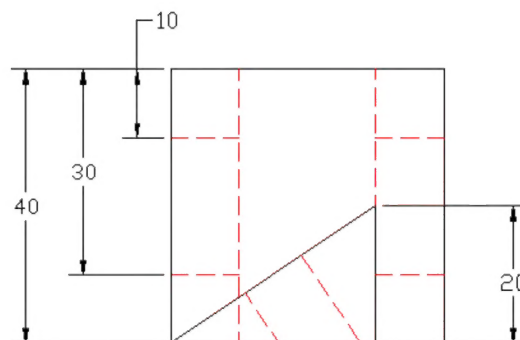
**STEP 6****Diameter and Radius Dimension**

Select **Diameter button**;  or **Radius button**;  under the **DIMENSION Menu** and click on the desired arc or circle. Look at **Figure 6.17** to see the placement of this dimension.

**Figure 6.17****STEP 7****Moving Dimension Number To A New Location**

Look at Figure 6.18. Notice that the position of the **10** for the slot in the bottom of this view has been changed from its original position.

To move this text, click on the **Override button**  on the **Dimension toolbar**. Click on the **10**, and use your LEFT mouse button to slide it to the new location.

**Figure 6.18**

**CONGRATULATIONS!!!**

You have completed the LAST AutoCAD Tutorial for this course.  
Good Job 😊.